Master’s Programme in Mechanical Engineering

Quasi-dynamic global strength analysis of a passenger ship in regular waves

Nikita Dementyev

Aalto University

MASTER’S THESIS

Aalto University
MASTER'S THESIS 2019

Quasi-dynamic global strength analysis of a passenger ship in regular waves

Nikita Dementyev

Thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Technology.
Otaniemi, 14 Oct 2019

Supervisor: Professor Spyros Hirdaris
Advisor: M.Sc. Markus Jokinen

Aalto University
School of Science
Master's Programme in Mechanical Engineering

Author
Nikita Dementyev

Title of thesis
Quasi-dynamic global strength analysis of a passenger ship in regular waves

Master programme Mechanical Engineering **Code** ENG25

Thesis supervisor Professor Spyros Hirdaris

Thesis advisor M.Sc. Markus Jokinen

Date 14 Oct 2019 **Pages** 59 + 19 **Language** English

Abstract

The current boom in cruise and passenger ship markets has led to a corresponding increase in the size of ships and their structural complexity. However, the optimization of capital expenditure costs remains a critical part in the design and construction of such ships. Additionally, the designers have at their disposal state-of-the-art tools and rational design methods for design and structural strength assessment of ships to ensure sufficient functional safety margins, especially for ships with general particulars and structural features that are not covered by the existing empirical Classification Rules.

This master's thesis presents a rational quasi-dynamic response approach for the evaluation of global loads of passenger vessels. A rational quasi-dynamic response method couples wave-induced hydrodynamic pressures with a rigid hull idealization performed with ANSYS AQWA and ANSYS SpaceClaim. The CAD structural model of a typical cruise ship was produced using CADMATIC Hull with basic design accuracy. Furthermore, it was sufficiently optimized in ANSYS SpaceClaim to obtain an FEA model comprising of shell elements representing the primary and secondary parts of the structure. NAPA software was used for evaluating the still water bending moment. Consequently, the 3D diffraction/radiation panel code ANSYS AQWA was used to define the wave pressures acting on the hull and loads were mapped on the hull surface and transferred to the ANSYS FEM solver for hydro-structure coupling. As a result, still water and wave bending moments are received as well as the ship's response.

Comparisons against Class Society Rule wave bending moment amidships demonstrates that the direct evaluation of the wave bending moment and shear force envelopes along the hull girder may be a preferred rational approach in terms of assuring global structural strength and optimizing total steel weight.

The outcomes of the thesis were presented during the Baltic Seas International Maritime Conference on 24.09.2019

Keywords passenger ship, ship's global strength, ship design, FEM, quasi-dynamic

Preface

This thesis work was carried out as a cooperation between Aalto University and Elomatic Oy. The outcomes of the thesis were presented during the Baltic Seas International Maritime Conference on 24.09.2019.

I would like to thank Elomatic for helping me with the thesis topic search and funding my thesis. Special thanks to Antti Yrjänäinen for his advisory during the thesis process and Markus Jokinen for his involment in supervising the thesis.

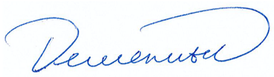
Exceptional gratitude to Professor Spyros Hirdaris for his guidance on the whole thesis and encouraging me to present the outcomes of this work during the Baltic Seas International Maritime Conference.

I wish to thank Laivanrakentajain kerho (LRK) of Aalto University for all the great moments and many friends I have met during my studies.

Finally, I would like to thank my lovely Diana for supporting me in the thesis process and my parents for encouraging me in my studies since day one.

Espoo 14.10.2019

Nikita Dementyev



Contents

List of Figures

List of Tables

Abbreviations

CAD Computer Aided Software.

CFD Computational Fluid Dynamics.

CSR Common Structural Rules.

DLP Dominant Load Parameter.

DNV GL Classification Society.

DOF Degree of Freedom.

FE Finite Element.

FEA Finite Element Analysis.

FEM Finite Element Method.

IACS International Assosiation of Class Societies.

RAO Common Structural Rules.

TEU Twenty-foot Equivalent Unit.

1. Introduction

This chapter gives an overview on the background of the thesis and presents its aims and limitations.

1.1 Background and motivation

Today more than 90% of the world global trade is moved by ships. This demand has drastically grown in the past 50 years. By the same time the size and complexity of ship structures has increased dramatically. This means that ship structural designers must use state-of-the-art tools and rational design methods when developing new vessels.Thus, main task of ship structural design is to ensure the safety of the vessel during the operational process, while also being cost-efficient and having the lowest lightweight.

In the past, ships were mainly designed based on the rules set by Classification Societies. These rules were mostly supported by empirical formulations and feedback from the ship operators' experience. Only using Classification Society rules, however, may be not suitable in the case of modern vessels, such as ultra-large container ships, liquefied gas carriers with a large cargo hold, or cruise ships with big promenades and open spaces. With the development of the finite element method, it is possible to conduct direct analysis and solve various complex structural problems of modern ship designs.

A major step in ship structural design is global strength analysis, which mainly aims to ensure the structural safety of the entire hull girder under bending and torsion in both still water and wave conditions. The global strength of the vessel is also usually referred as longitudinal strength, as mostly the longitudinal bending requires most attention. Figure 1.1 represents the global strength failure of a containership. In this case as a result of the harsh weather, extensive wave loading caused the ship's hull to split and then finally break in two. The wave loading exerted on the ship exceeded the maximum ultimate strength of the vessel, which led firstly to a crack in the hull structure and then finally to the splitting of the vessel. [13].

Figure 1.1. A container ship broken in half [2]

In order to avoid such catastrophes and to ensure adequate longitudinal strength, global strength analysis is performed. Such analysis typically consists of still water condition and wave condition. In the still water condition, the moments acting on the vessel in completely calm water are considered. This problem is of a hydrostatic kind and mainly depends on the vessels loading condition and buoyancy forces acting on it. Most of the still water condition related issues are checked within the hydrostatic software or class society software, that utilizes a ship's basic data.

From a modelling and simulation perspective the idealization of a realistic wave condition usually requires significant knowledge and effort. This is the reason why empirical rules are used for global strength assessment [14]. For ships of abnormal configuration and complexity Classification Societies require the application quasi static, hydrodynamic or hydroelastic modelling procedures [15].

1.2 Aim of the thesis

The main aim of this thesis is to study the application of modern quasi-dynamic methods of global strength analysis using state of the art commercial software. A rational methodology is described for use in the overall ship design process.

Current class society rules do not provide the extended regulation on cruise ships longitudinal strength. For this reason a comparison between the class society rules on the bending moment (envelope curves) and actual bending moment from the 3D global model hydrodynamic analysis will be the one of the primary goals of this thesis work.

1.3 Scope of the work

There are certain limitations, that define the scope of the thesis:

- This thesis concentrates on the holistic process of ship global strength analysis. Focus is then set to the conceptual and basic ship design levels. This means that the most important major details related with early ship design process. Accordingly, the details related to the detail design steps can be only mentioned or disregarded.
- There are several major aspects in global strength estimation process. Various tools are used for each of the steps. However, the focus is set on global strength analysis using the finite element method.
- Load application is an important part of global strength. Therefore, the loading method must be relevant to represent the actual sea state. However, complex hydrodynamic models requires a large amount of computational power.
- The process of global model creation is described in this thesis. It begins with the 3D model of the vessel made in CAD software and all the transferring and modifications for global strength analysis.

2. State of the art

This chapter reviews modern methods of ship's global strength analysis. Some of the latest research articles are reviewed and modern software packages are presented.

2.1 Global strength analysis procedure

2.1.1 Global finite element method

With the current development of modern computational methods, global strength analysis is now possible. Typically, such an approach requires deterministic estimation of the hull strength. This is usually done with the help of computational hydrodynamic tools for hydrodynamic loading and FEM for structural response analysis [16].

Figure 2.1 represents an approach to the holistic ship structural design presented by Hughes and Payer [3], where the four first steps refer to the global strength evaluation. One of the most important parts of the whole process is the specification of loads acting on the vessel. This can be done in a way of applying the static loads, representing the maximum moments suggested by class rules, or by a more sophisticated method that models the hydrodynamic loads acting on the vessel. The complexity of hydrodynamic modelling varies depending on the taxonomy of the method used for the implementation of the influence of fluid structure interaction on dynamic response [17].

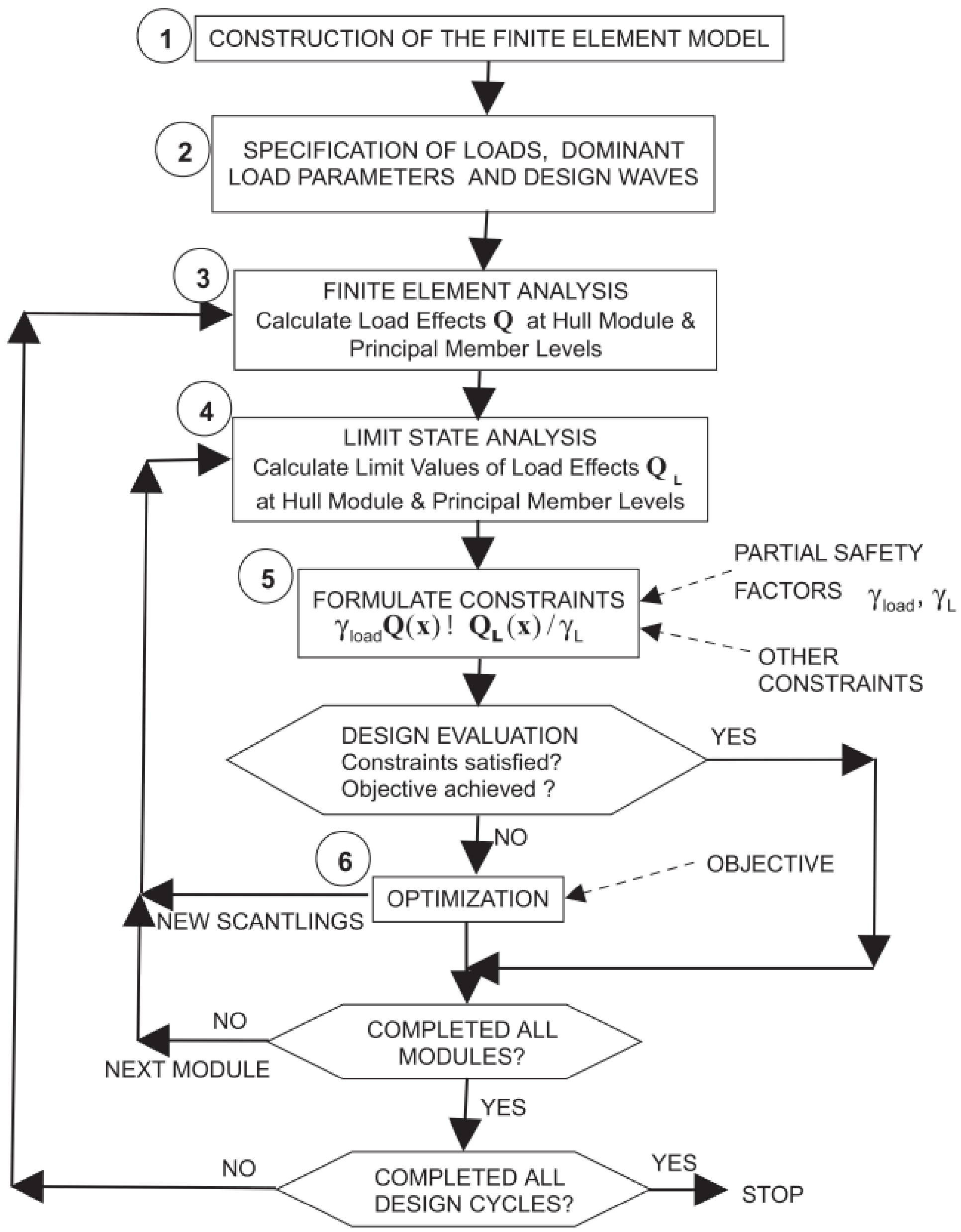


Figure 2.1. An example approach to the ship structural design [3]

The global finite element model typically represents the whole vessel's structure with a low level of detail and mainly load carrying constructions modelled. The model is then discretized with a coarse mesh ranging in size from the frame spacing to 3 meters [18].

Global finite element models are very demanding to create as they involve a lot of manual modelling hours. This includes model creation or transferring, meshing and balancing with the hydrodynamic mode. The accuracy of the model must be suitable for both lower computational time and its utilization in the future. A good global finite element model serves then as a base for the fast local model creation and verification and accelerates the design process. It also is necessary to perform the global strength analysis. Preliminary bending moment calculations are done at the concept stage. At the basic design stage, the global finite element analysis is done using the appropriate loading. Finally, at the detail design stage, the model can be modified and updated.

2.1.2 Loads and dynamic response

As the starting point of longitudinal strength, typically the still water condition is checked. This requires estimation of the still water bending moment and shear force that are produced by integrating the net load curve, or the difference between the ship weight distribution against the buoyancy along the vessel's length. Today this is mostly done using computer software. Such approach gives the distribution of forces based on input hull form and loading conditions. Ship theory software, such as NAPA, is capable of carrying out such calculations. Classification Societies also provide computer programs for verification of scantlings.

There are typically two types of ship response in waves based on the natural frequency of the applied loading. Those are known as quasi -static, quasi-dynamic and hydroelastic. Hydroelastic responses are considered beyond the purpose and objectives of this thesis and thereto are not considered.

In quasi-static response the load on the vessel is assumed to be slowly applied, or in other words almost static having no acceleration. In this case the response of the ship is mostly affected by the stiffness of her structure, while the inertia effects are relatively minor and can be completely neglected from the model. The actual situation usually is that the loading is not static and entails some acceleration, but in relation to the ship's size and motions, it is assumed static. Quasi-static analysis is independent of time when loads and deflections vary linearly at each location of the vessel. Global strength also complies with this category as the deformations of steel structures under the global ship loads are mainly the goal of whole ship model analysis [3].

Dynamic response analysis, on the other hand, is an investigation where the loads on the ship are applied at a high frequency that may influence global strength. In this case, the response of the vessel is mostly affected by the inertia forces, while its stiffness is not taken into account. Dynamic loads are time-dependent and may cause heavy vibrations and possible fracture of the structure. That is why such events as bow slamming, whipping and springing of the whole ship, are mostly studied using the dynamic response analysis [19]. A more detailed description of the response analysis type is given in chapter 3.

2.2 Classification Society regulations

Class Societies develop rules, regulations and classification guidance notes for the evaluation of ship's global strength. Those regulations are based on empirical equations that consider the ship's main dimensions, which then give the limit curves for the global bending moments and shear forces. This formulation for the limit curves in the class society rules, such as DNV GL, are usually based on the standard by International Association of Class Societies (DNV GL) UR S11A [20]. This standard provides requirements for the longitudinal strength of vessels with a length greater than ninety meters, with special attention on the bulk carriers, container ships, and oil tankers.

With the latest major development of passenger ship market, the need for safe structural passenger ship designs becomes eminent. Therefore, Classification Societies have developed special guidelines for direct analysis of wave loads on passenger ships. Modern cruise ships and ferries have massive superstructure spreading at the height of few decks. This structure in many cases may contribute on the augmentations of ship's global response. This augmentation of the superstructure's load carrying mechanism is usually one of the main reasons leading to the application of direct strength analysis procedures. For example, DNV GL suggests a guideline for the direct analysis of passenger ships in addition to their guidelines on global FEM analysis [21]. Another Classification Society, Lloyd's Register, provides a detailed procedure for the direct analysis of passenger ships using finite elements, where special attention is given to the global and double bottom strength verification [22]. The main requirements for performing the detailed procedure are once again on the utilization of superstructure in the global load carrying system and innovative structural design.

2.3 Literature review

Ship's global strength analysis is a typical routine for the bulk carriers or container ships with a free cargo hold. An article by Malenica and Derbanne from Bureau Veritas explains the process of hydro-structure interaction for ultra large container ships [19]. The authors describe the procedure starting from the hydrodynamic and FEM calculations and with special attention address the transfer of hydrodynamic loads to the FEM model. Different hydro-structure interaction models, represented in table 2.1, are described from the structural side, as quasi-static and dynamic, and from the hydrodynamic side, as linear, wave non-linear and impulsive non-linear. Authors then state that even without taking the hydroelastic effects into account, the classical quasi-static hydro-structure interaction is not fully mastered nowadays and is not included in the design procedure on an acceptable level.

Table 2.1. Hydro-structure interaction methods

	Linear	**Wave non-linear**	**Impulsive non-linear**
Quasi-static	✓	✓	✓
Dynamic	✓	✓	✓

Estimation of the hydrodynamic loading on the vessel is one of the most important issues in the ship structural design. An article by Hirdaris et. al. provides an overview of the modern prediction methods of the wave-induced loads on ships and offshore structures [14]. In this work, the forward speed method is typically classified into six levels, depending on the complexity and computational time:

- Linear
- Freude-Krylov non-linear
- Body non-linear
- Body exact – weak scatterer
- Full non-linear - Smooth waves
- Fully non-linear

Industrial application using commercial software mostly relies on level 1 (linear) and level 2 (Froude-Krylov non-linear) methods. In linear cases, the ship's wetted surface is defined by the mean position of the vessel's hull under the respective wave condition. The solution is then provided in the frequency domain by pulsating source methods. In Froude-Krylov non-linear methods the hydrostatic and wave-induced pressures are instantaneously estimated over the wetted surface.

As the coupling of hydro-structure interaction usually requires some kind of integration between the FEM solver and ship hydrodynamic software, Classification Societies tend to develop their packages enabling the coupling of FEM and hydrodynamics for the research and classification needs. One of such examples is the GL ShipLoad developed in the mid-2000s [23]. In this type of software, application of a global FE model was used as input for the software. The equivalent design wave approach is then implemented by applying a set, chosen from the harmonic waves, of designed waves which are applicable for the current vessel and its operational area. Those design waves were then used to produce sectional loads on the ship's hull. Dominant load parameters (DLP) are referred to the most significant global loading case or motion in the FEM analysis. DLP are specified by the Classification Society based on the ship type and allow calculation of the most important loading saving the computational time. For the case of a container ship, the DLP was positioned in way of the midship hogging condition under head seas [23].

A view of the class society on the rational ship structural design is described in the article by Payer[24]. The authors state that a pure rule book approach in ship structural design may lead to certain risks and disadvantages associated with the complexity and random character of ship failure modes. As the starting point, the authors highlight the importance of wave-induced global hull loads resulting from the mapping of hydrodynamic pressures. The case study of this article is a 13 000 TEU container ship, for which the finite element model was subject to hydrodynamic loading obtained from a linear strip theory code with the addition of viscous roll damping. Correction for the non-linearities of the hydrodynamic pressures in the waves was also performed. [25]. As a result, the bending moments and shearing forces were obtained in addition with the ships motion accelerations.

Naar's doctoral dissertation on the ultimate strength of the large passenger ship utilizes non-linear coupled beam theory and FEM [26]. The study concentrates on the estimation of ultimate strength of the prismatic midship section for the case of a post-Panamax size passenger ship. The hull structure was modelled using the beam elements representing bulkheads, decks and side structure and non-linear springs to simulate the coupling between these beam elements. Although the main target of this work was to develop an ultimate strength estimation method, it was elaborated that deflections and stresses in the hull structure may also be estimated using the non-linear coupled beam theory. The study was held on the ultimate strength level, i.e maximum strength of the hull. Therefore, there was no hydrodynamic loading used in this work. However major FEM related issues were described, such as modelling and meshing of the structure, and post-processing.

2.4 Recent tools for global strength analysis

2.4.1 Ship design dedicated software

There are various ship design tools used in the process of global strength analysis. Some of that software is not developed especially for ship design but is used for common engineering purposes or with the help of special modules it becomes suitable for ship specific applications.

In these software packages usually a 3D model of the vessel is constructed and the complete design process is based on it. Conceptual, basic and sometimes even detail design steps can be done in one integrated environment. Such software packages are for example NAPA, AVEVA and Foran [27]. AVEVA is now the leading software in the field of ship structural design [28]. NAPA has extensive features dedicated to ship theory. NAPA allows creation and fairing of hull surfaces. Accordingly, rooms and compartments can be defined for the initial general arrangement with the help of geometry editor. In the case of global strength estimation, NAPA can calculate hydrostatics, bending moments along the hull for both still water and wave condition, which can then be compared to the class society's values. NAPA steel module allows creation of the vessel's structural model and its further conversion to the FEM model with meshing inside NAPA [29].

CADMATIC software offers a 3D model based ship design solution for most of the design steps. Marine solutions consist of two main parts: CADMATIC Hull and Outfitting. CADMATIC Hull allows flexible construction of the structural 3D model, which is valid for conceptual, basic, detail design and production drawings [30]. The main principle of the Hull package is that the element's creation mainly is done in 2D section drawings, while some modifications can be done in the 3D model, which is automatically updated with the created parts. This allows creating a base for production drawings already at conceptual stage. Model creation is typically done using the standard shipbuilding parts: plates, shell plates, profiles, brackets, pillars, etc. Any kind of modification is then possible to do within the software and the whole 3D model is connected with the outfitting model and updated respectively. Hull Viewer is the internal software which is used for viewing the 3D model in CADMATIC. It is also directly connected to the Hull and for example selection of items can be done in Hull Viewer for any modification in Hull module. After the steel model was created it can be shared with a common CAD file export standard to any other CAD software. One of the specific features of CADMATIC is integration with Bureau Veritas class software, where the transversal sections created in CADMATIC can be shared with the BV class software.

2.4.2 Class societies scantlings software

Class societies have a long history of developing software for both their own needs and for the needs of their customers. The software application area then greatly varies with in-house packages available for basically any ship design and operational discipline. One kind of such software is the tool for checking the scantlings of the vessel according to the rules. Some examples can be Mars2000 by Bureau Veritas, Nauticus Hull by DNV GL and RulesCalc by Lloyd's register [31], [32], [33]. The main idea behind such software is to design and optimize the scantlings of the vessel according to the Classification Society rules. Typically, there is an option to select current class rules or for example IACS rules for bulk carriers and oil tankers. Various integrations to CAD or FEA is also possible within the packages. Transversal sections are usually the main input for the software and most of the time it is only the midship section to be verified, which brings the main drawback of such a solution. The vessel is then idealized based on the midship section, thus the correct strength distribution along the longitudinal direction can be completely different from the real vessel, due to the special structural features of the hull.

2.4.3 Hydrodynamic software

Hydrodynamic software focuses on the prediction of motions and loads in waves. Such software can estimate the ship motions, pressures exerted on the wetted surface, wave loads , response amplitude operators for wave motions, multi-body interaction in waves, etc. Typically, the software is based on one of the seakeeping codes and can work in frequency or time domains.

One of such software examples is ANSYS AQWA. It is mainly based on the hydrodynamic diffraction and vastly used in the offshore industry for calculating the motions on the platforms. However, it also includes radiation features, so complete analysis of wave loads on a ship can be performed. One of the advantages of AQWA is that it can directly couple wave loads with the structural model for the FEA. The software programs published by the Classification Societies are: FD Waveload by Lloyd's Register and Hydrostar by Bureau Veritas [34], [35]. While their functionality might be very similar to Ansys AQWA, the benefit of such solutions is that the software was developed in-house specifically for ship design and classification needs and has a long history of usage, also verified by experimental data.

2.4.4 Finite element software

The finite element method was developed in the 1940s, while its major development was done in the second half of the twentieth century [3]. That allowed ship designs to be based not only on the empirical rule formulas but also to verify the ship structure by simulating its behavior using finite elements. Nowadays, there is a large amount of FEM software as well as for general industrial applications and some specific applications and needs. A few of the largest and best-known software packages presented today on the market and used in the shipbuilding industry are Ansys, Abaqus, and Femap [9], [36], [37]. Class societies have developed purpose made software for classification and design optimization. Some examples are Shipright by Lloyd's Register and DNV GL Poseidon [38], [39]. The main advantages of modern commercial FEM software are easy to use graphical interface, tools for both pre- and post-processing. 3D models can be imported from any CAD software for meshing and post-processing.

3. Methodology

The current chapter represents the methodology used for this thesis. The theoretical background is presented as a foundation for practical application in the following chapters. The chapter concentrates on the ship loading, analysis types and then ship response. An overview of the hydrodynamic force acting on a vessel in waves is also given.

3.1 Loads on ships

3.1.1 Loading types

When dealing with the ship strength analysis it is important to first identify the loads which are acting on the vessel. One way to classify the loads on the vessel is according to their variation in time. Loads that do not vary in time are static. Loads that vary in time can be slowly varying or rapidly varying [3]. There are also corresponding types of structural analysis, which are mainly defined by the approximation of applied loading: static, quasi-static and dynamic [3]. Nowadays in ship structural design it can also be defined as quasi-static, quasi-dynamic and dynamic [15].

Another important issue in the load effect estimation is the domain in which the problem is solved: frequency or time. If displacement varies linearly or non-linearly with the applied loading it is preferably solved in the frequency domain. The loading or response distribution in the frequency domain is called a spectrum. If the displacement varies non-linearly with the loading then the problem should be solved in the time domain. However, complex non-linear problems can be usually decomposed into several linear problems, that may be treated in the frequency domain.

Due to the nature of water and waves, there are a few loads that may be considered as completely static. Typical static loads on ships are:

- Still water loads: buoyancy and weights
- Drydocking loads
- Thermal loads

In the holistic ship structural design the most significant static loading is the still water loading and it especially important in the global strength analysis, as it is affecting the whole ship [3].

Slowly varying loads are defined then as the loads where the shortest loading period is way larger than the structure's natural frequency. These loads are usually dealt with the use of the quasi-static analysis, or approximating the load as almost static, which introduces a small calculation error. Most typical loads of this kind are:

- Wave-induced hydrodynamic pressure on the hull
- Sloshing in tanks and cargo
- Green water on deck
- Wave slaps on the vessel
- Inertia loads on the heavy objects
- Ice-breaking loads

The most remarkable loading from the slowly varying type acting on all types of vessels is the wave-induced hydrodynamic pressure, which is caused by the accumulation of wave load and the consequent ship motion [3].

Finally, rapidly varying loads are loads of short period, or significantly shorter than the structure's natural period. They usually require dynamic analysis and time domain simulation. Typical rapidly varying loads are:

- Slamming
- Forced vibrations (machinery and propulsion)
- Springing and whipping

In the rapidly varying loads, there are loads, which affect the vessel much more than others in any situation. These loads should be considered separately using the dynamic analysis.

3.1.2 Still water condition

While a ship is in the water she becomes subject to a still water bending moment and shear force acting on her at any time of the operation. Assuming ideal still water conditions, the area of the wetted surface will remain constant. Shear force and still water bending moment typically arise from two kinds of loads, which are the displacement or total weight of the ship and the buoyancy of water applied to the submerged hull. The main forces acting on the ship are represented in Figure 3.1 [4].

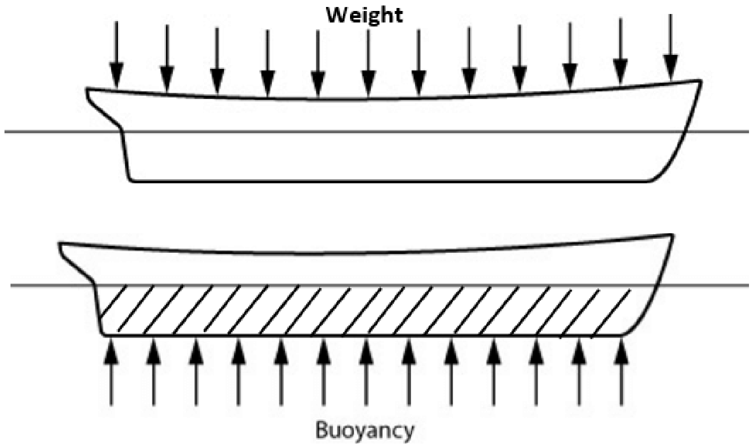


Figure 3.1. Major forces acting on the vessel in still water [4]

The ship's displacement consists of two main components: lightweight and deadweight. Lightweight or lightship is the own weight of the ship, which consists of hull structures, machinery, and outfitting. This weight is estimated during the design process, measured in the construction and stays more or less constant during the lifetime of the ship when any modification to the lightship is updated to the vessel's information. Deadweight, on the other hand, consists of the weight of the cargo, passengers and crew, consumables on the vessel, such as fuel oil, lubricating oil, spare parts, ballast, and freshwater, etc. The deadweight does not remain constant during the operation and the distribution of the deadweight may be completely different, for example with the movement of passengers along the vessel. This is usually approximated using the loading conditions, i.e. various distribution of cargo loads and ballast conditions, in the basic hydrostatic calculations [40].

Buoyancy is the upward force that is exerted on the ship's underwater part of the hull. In the equilibrium condition, the total sum of buoyant force and the vessel's displacement is zero, which keeps the ship afloat at even trim. However, due to the fact of uneven weight distribution along the ship and the variation of buoyant force along the hull form, the local force at each point along the vessel may not be zero, which creates the still water shear force. At each section of the hull, this shear force may be zero or point either upwards or downwards, which will create respective still water bending moment.

Typically, the net loading curve, p, is drawn by subjecting the buoyancy curve from the weight distribution curve. Then by integrating the net loading curve the distribution of shear force on the ship's hull is obtained, $F_S = \int pdx$. A double integration of the net loading curve will provide the still water bending moment, $M_S = \int F_S dx = \int \int pdxdx$. Figure 3.2 represents a typical example of the net loading, shear force and still water bending moment [5].

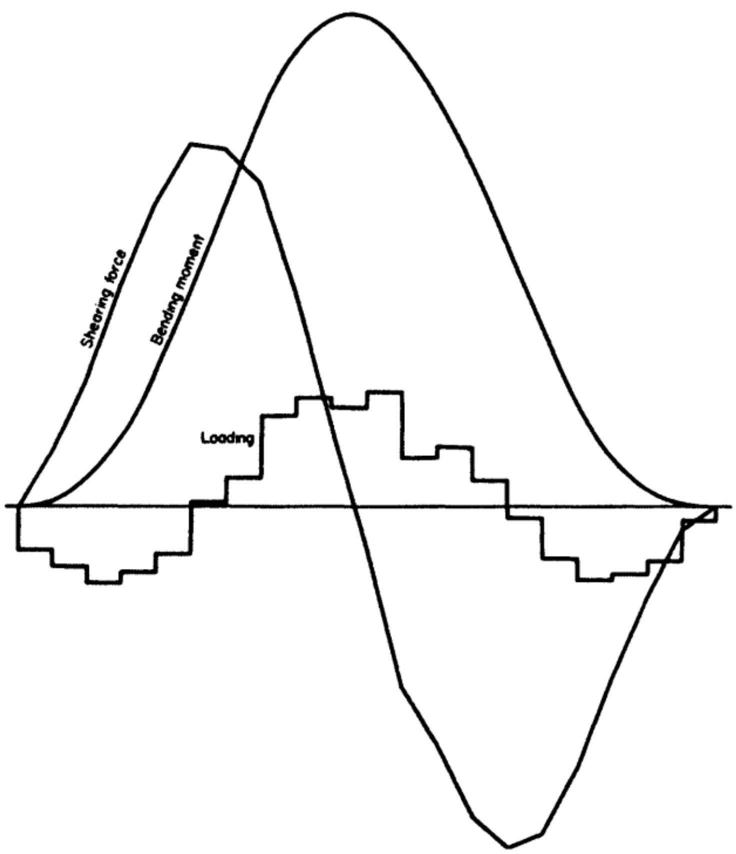


Figure 3.2. An example of net loading, shear force and still water bending moment [5]

The still water shear force and bending moment are possible to obtain using the basic strip theory or hydrostatics software [29]. Initial data on the hull form and weight distribution is of course required for such calculations. Then still water curves can be plotted. Class societies' software can also provide the still water curves based on the ship initial data. Usually, this step of longitudinal strength calculation is done at the conceptual stage of the design. The hull form and the design draft are defined in the buoyancy calculation. The weight distribution is typically obtained using CAD software, especially for the steel weight, however, weight tables based on the references and statistics are applied as well for all the weight components.

3.1.3 Hogging and sagging condition

In the longitudinal strength analysis, the maximum values of the bending moment are usually of the most interest to the ship designers. The maximum bending moment results in the corresponding maximum value of normal stress at the midship, which may is the most likely to lead to a structural failure. There are two main types of global hull loading, depending on the curvature of the bent hull it causes: sagging and hogging. Figure 3.3 represents the curvature of the hull bending under the corresponding global loading.

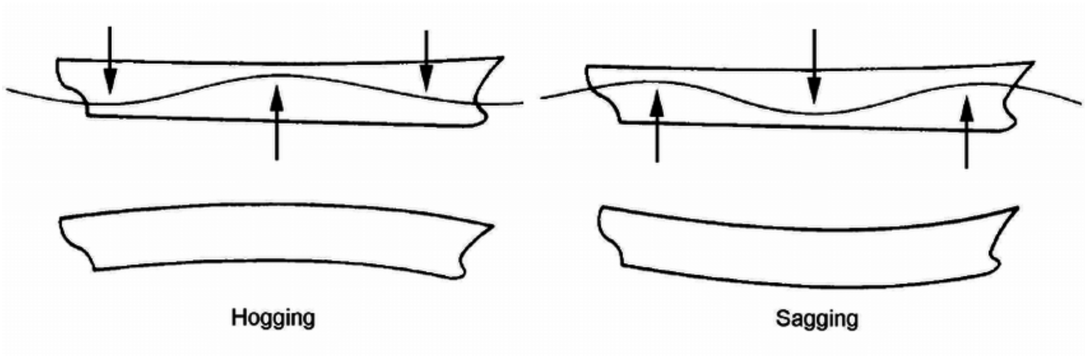


Figure 3.3. Hogging and sagging hull bending

In hogging the buoyancy force is concentrated on the midship, where the bow and the stern of the vessel are pushed down by the gravitational force. Sagging represents an opposite situation, where the buoyancy is shifted towards the end of the vessel, while the midship is pulled downwards. Tension and compression stresses are acting on the deck and bottom of the ship when it is sagging or hogging.

Hogging and sagging conditions can be caused by different issues. Cargo distribution along the vessel can be uneven which leads to a ship's hull bending in form of one of the curves. If the cargo is loaded unevenly on the vessel's cargo hold it can cause the vessel to hog if the cargo is loaded at the end and to sag if the cargo is loaded in the midship area. This may lead to the vessel breaking in two even in the still water when loading the cargo. Therefore, cargo must be evenly distributed in the holds.

Another situation causing the vessel to hog or sag is when the vessel is in waves. If the wave crest is at bow and stern of the vessel, the gravitational force pulls the midship downwards making the ship to sag. In the hogging condition wave crest is at the midship and the ends of the vessel are pulled downwards.

One of the important issues related to the hogging and sagging moments is the difference in the prediction of the moments using the linear wave theories. Hogging and sagging moments are typically different in their modulus due to various factors, such as hull geometry of the vessel at bow and stern, weight distribution, etc. Linear wave theories are unable to consider those details, and thus unable to predict the difference in the distribution of the sagging and hogging moments [41].

3.1.4 Regular waves

Linear regular waves are the simplest ocean wave elements are used to model the conditions of the sea state. There are approximations from simple trochoidal or sinusoidal wave assumptions, which are acknowledged to have permanent shape and parameters along the length of the wave. Such waves can represent the actual characteristics of the sea state, however, they should be considered as only an approximation of the complex real wave [5].

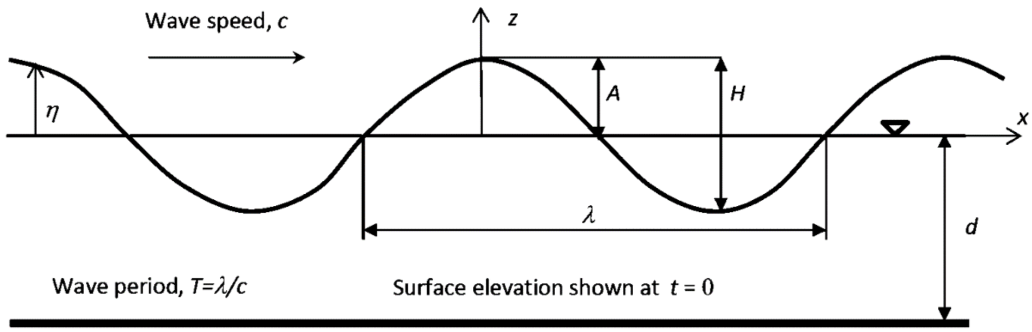


Figure 3.4. A typical representation of a regular wave [6]

Figure 3.4 represents the profile of a typical regular wave. A regular wave can be characterized by some of its main parameters: wave length, λ, in m, wave period, T, in s, phase velocity, c, in m/s, wave frequency, f, in Hz, wave height, H, in m, wave amplitude, A, in m and surface elevation, η, in m [6].

The wave length of a regular wave can be estimated as:

$$\lambda = \frac{g}{2\pi} \cdot T^2 = 1.56T^2 = 1.56/f^2 \tag{3.1}$$

The wave speed of the propagating crest can be found as:

$$c = \frac{\lambda}{T} = \frac{g \cdot T}{2\pi} = 1.56T = 1.56/f \tag{3.2}$$

3.2 Analysis types

3.2.1 Quasi-static analysis

When the ship is moving in waves, both ship and water experience accelerations in all six degrees of freedom. However, solving the coupled problem of hydrodynamic loading and ship response in the time domain is extremely demanding, especially considering that dynamic loading is changing in time. Thus, quasi-static response analysis is introduced, where the loading on the vessel is assumed to be almost static, or very slow. In this case, the total stiffness of the vessel affects response and inertia effects can be neglected [19].

Another major difference between the quasi-static and dynamic response analyses is that the vibrations are not taken into account in the quasi-static analysis, as the ship is considered rigid and the stiffness of the ship is affecting the response. Assuming steady state conditions wave loading applies slowly, thus the structure deforms on a slow strain rate with mostly elastic deformations. Typically, in quasi-static analysis, the loading is modelled by using the trochoidal wave distributed along the vessel. Forces, arising from that wave, can be then applied at the ends of the vessel statically.

Due to the nature of wave loading, there is always a need for the idealization of applied loading. Moreover, an approximation of the dynamic loading at each step of the time in the quasi-static approach also leads to uncertainties.

3.2.2 Dynamic analysis

In the dynamic response analysis, the main focus is given to the inertia effects and the vibration of the structure, while the structural stiffness does not play a major role.

Rapidly varying loads typically arise from impact wave loading or the matching wave encounter frequency which results in heavy structural vibrations. To accurately simulate vibrations the vessel is modelled as an elastic structure. It is believed that in certain cases more advanced hydroleastic analysis may help to idealize the time dependent integration of hydrodynamic loads and elastic structural deformations within the context of fluid structure interactions [42].

3.2.3 Quasi-dynamic analysis

Several challenges occur when using quasi-static response analysis for dynamic load modelling. This is usually related to the influence of non-linearities and several uncertainties introduced by them in the computation of the dynamic wave loads [14]. To overcome the limitation associated with the modelling of wave loads in the quasi-static response computational methods, such as 3D linear seakeeping or weakly non-linear hydrodynamics were introduced. Such a method is sometimes called quasi-dynamic response analysis, as it utilizes some features of the quasi-static and dynamic analysis [16]. Table 3.1 represents the comparison between the quasi-static and quasi-dynamic methods.

Table 3.1. Comparison of quasi-static and quasi-dynamic analyses

	Quasi-static	**Quasi-dynamic**
Hydrodynamic loading	Static (Point load)	Dynamic (Hydrodynamic pressure)
Vessel in waves	Rigid	Rigid
Inertia effects	$\times$	✓
Vibration effects	$\times$	$\times$
Structural analysis	Static	Static

The main advantage of this method is that the wave loading is approximated dynamically as hydrodynamic pressures exerted on a hull surface due to the approached wave. Strength analysis, in this case, is done the same way as in the quasi-static analysis, where the maximum loading resulting from a dynamic wave at a point in time is considered. The loading is then applied statically, but in contrast to the quasi-static analysis, where the loading is based on the statistics of ship data and applied as point loads on the rigid hull, the quasi-dynamic analysis considers a dynamic loading, which is applied as the hydrodynamic pressures on the hull. [14].

3.3 Hydrodynamic forces

According to the hydrodynamic principles, the forces acting on a vessel are typically subdivided into three categories: hydrostatic, radiation and diffraction forces. The derivation of such division can be obtained using potential flow theory. According to the potential flow theory the total potential of the forces acting on the vessel is:

$$\phi(x, y, z, t) = \phi_I + \phi_D + \phi_R \tag{3.3}$$

Where the first term ϕ_I represents the incident wave potential of the waves omitted on the vessel. The second term, ϕ_D, is potential which accounts for the diffracted waves resulting from the contact of the vessel with the body represents the wave diffraction problem. And the third term, ϕ_R, is velocity potential of the ship oscillations in the absence of incident waves, which represents the radiation problem. In the linear theory, all of the above problems are assumed to be independent of each other, if the ship motions are small-scale [43].

Using the Laplace and Bernoulli equations the total force acting on the vessel can be later expressed as:

$$\vec{F} = \vec{F}_{HS} + \vec{F}_{HSW} + \vec{F}_{RAD} + \vec{F}_{HYD} \tag{3.4}$$

where the four components in the equation are:

- $\vec{F}_{HS}$ is the hydrostatic component acting on the ship in still water condition
- $\vec{F}_{HSW}$ is the hydrostatic component due to the existence of waves
- $\vec{F}_{RAD}$ is the radiation force containing added mass and damping components
- $\vec{F}_{HYD}$ are the hydrodynamic excitation forces

The first component was described in Chapter 3.1.2 and represents purely hydrostatic forces acting on a vessel in still water condition. The second component describes the hydrostatic forces arising due to waves acting on the varying hull wetted surface. This force is also of a hydrostatic kind, but instead of a still water condition, it takes into account the hydrostatic pressure of the incident waves [43].

3.3.1 Radiation force

The third component of the equation represents wave inertia force or radiation force which is caused by waves created from the presence of the vessel in the water. Incident waves are not taken into account in the radiation force, thus the vessel is assumed to be oscillating in the still water. The fluid potential theory is usually used for solving the wave inertia forces and wave exciting force. Figure 3.5 illustrates the body in water oscillating with the velocity $\vec{U}$ and radiation potentials ϕ_R.

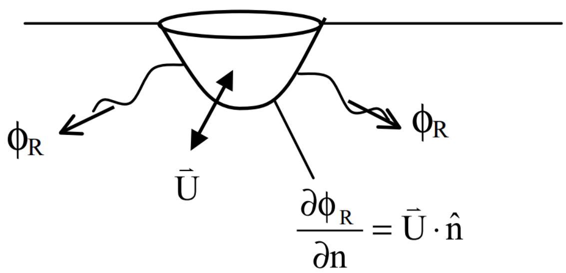


Figure 3.5. Radiation force of a body oscillating in waves [7]

According to the potential flow theory radiation force can be represented as:

$$\vec{F_R} = \iint\limits_{\substack{\text{wetted}\\\text{surface}}} -\rho \frac{\delta \phi_R}{\delta t} \hat{n} dS = -M_{ij}\dot{U}_j - D_{ij}U_j \tag{3.5}$$

where ρ is the water density, S is the wetted surface of the vessel, M_{ij} is the added mass coefficient and D_{ij} is the damping coefficient.

3.3.2 Added mass

The force acting on an oscillating body always takes into account the term associated with the added mass. When the ship moves in waves there is some amount of water which travels along with the vessel. At each frame of the vessel, there is a certain mass of water, which must be taken into account during the seakeeping calculation.

In low frequency waves the added mass goes towards zero, but at high frequency waves, it approaches a constant value. For ships, the added mass can reach one fourth or one third of the total ship weight. This added mass creates a large inertia force [6].

THe added mass coefficient of a ship in waves can be estimated using seakeeping methods. In strip theory the added mass of a vessel can be estimated by integrating the added mass of at frames along the length of the vessel [43]:

$$M_{ij} = \int_{-L/2}^{L/2} m_{ij} dx \tag{3.6}$$

3.3.3 Hydrodynamic damping

The decay of the ship motions in still water is caused by the hydrodynamic damping. Two main reasons are causing hydrodynamic damping of a ship. The first cause is the viscous damping, which is proportional to the velocity of the vessel. In the linear potential theory it is neglected as its contributions to total damping are small.

The second important term is damping caused by the radiated waves and is based on the energy transfer principle. When the vessel is oscillating in the still water it is generating radiation waves, that have both potential and kinetic energy. The radiation waves then transfer the kinetic energy of the vessel further from it and, therefore, decrease the total amount of kinetic energy. Thus, in time the kinetic energy decreases and oscillations of the vessel decay.

Similar to the added mass, radiation damping coefficient can be found by integrating the damping coefficients at ship frames along the whole length of the vessel:

$$D_{ij} = \int_{-L/2}^{L/2} d_{ij} dx \tag{3.7}$$

However, the linear potential theory does not account for viscous damping and can overestimate the roll amplitude in resonance. Thus, it is important to note the value of roll damping, that strictly depends on the viscous effects. Estimation methods based on the empirical and experimental data are used to roll damping and viscosity effects [44].

Various terms are affecting roll damping estimation. The first, is the wave making component, which can make up to 5-30 % of the total ship's roll damping. It must be estimated for both zero speed case using strip theory and forward speed case using mote complex methods [44].

During the sway motion, there are lifting forces acting on the hull. These forces may cause roll motions and therefore they affect the roll damping as the hull lift component. Frictional forces acting between the hull and water can make up to 8-10 % of the total roll damping. Empirical values are used for the estimation of the frictional component based on the damping of rolling cylinders [44].

The next component of the roll damping is the eddy making component. It arises from the sectional vortices generated during the roll motion. Eddy making component highly depends on the hull shape of the vessel and the theory is based on the experiments on the rotating Lewis cylinder. The eddy making component is decreasing with increasing forward speed [44].

The last component of the roll damping is the appendages component. Typically, those are bilge keel, fin stabilizers and skeg appendages. Assumptions behind these components are also based on the empirical data and experiments with and without the appendages [44].

3.3.4 Froude-Krylov and diffraction forces

When the radiation force considers the force arising from the vessel existing in still water, Froude-Krylov and diffraction forces make up so-called wave excitation forces, the fourth component of Equation 3.4, acting when the vessel is in regular waves.

The Froude-Krylov force considers the force created by the unsteady pressures made by the undisturbed waves. Also known as wave-induced force, the Froude-Krylov forces can be both hydrostatic and hydrodynamic kind. When approaching the vessel the induced wave field contains both hydrostatic force, caused by the change of the wetted surface due to waves and hydrodynamic force, caused by the wave inertia. The Froude-Krylov force can be estimated from [7]:

$$\vec{F}_{FK} = -\iint\limits_{\substack{\text{wetted}\\\text{surface}}} p\vec{n}dS \tag{3.8}$$

Where p is the pressure of the undisturbed waves. Worth noting that only the hydrostatic part of the Froude-Krylov force is represented in Equation 3.4 in the second component. Hydrostatic force is not a part of the fourth component of the equation, as only the hydrodynamic part of the Froude-Krylov force is included in the wave excitation force. After the induced wave meets the ship it is diffracted and moves away from it [43].

The second part of the wave-excitation force is the diffraction force. Figure 3.6 represents the diffracted wave potentials, which are the cause of the wave diffraction force. Unlike the wave-induced forces, the wave diffraction forces take into account the waves disturbed by the presence of the vessel in the water [43].

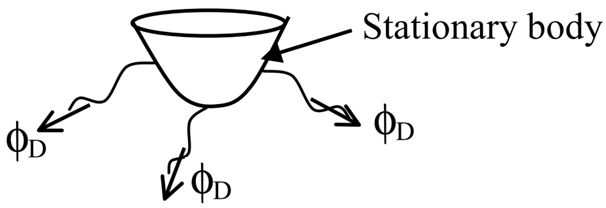


Figure 3.6. Diffraction forces acting on a body at rest [7]

The diffraction force can be estimated from:

$$\vec{F}_D = \iint\limits_{\substack{\text{wetted}\\\text{surface}}} -\rho\frac{\delta\phi_D}{\delta t}\hat{n}dS \tag{3.9}$$

When the wave field approaches a vessel energy of the incident wave is decreased. However, the change of potential of an incident wave is equal in the magnitude to the change of potential of a diffracted wave, $\frac{\delta\phi_I}{dt} = -\frac{\delta\phi_D}{dt}$ [7].

3.3.5 Response Amplitude Operators

In naval architecture, the term Response Amplitude Operator (RAO) represents behavior of the vessel. In ANSYS AQWA a set of linear equations is solved to get the harmonic response of the vessel in regular waves. RAO being the response characteristics is then proportional to wave-induced amplitude.

To obtain ship RAO, the ship equation of motion is needed:

$$\Big(- \omega_e^2 (M_s + M_a) - i\omega_e C + K_{hys} \Big) x_{jm} = F_{jm} \tag{3.10}$$

where ω_e is the encounter wave frequency, M_s is the structural mass, M_a is the added mass, C is the damping coefficient, K_{hys} is the hydrostatic force and F_{jm} is the wave excitation force consisting of diffraction and Froude-Krylov forces. Equation 3.10 becomes:

$$x_{jm} = HF_{jm} \tag{3.11}$$

where

$$H = \Big(- \omega_e^2 (M_s + M_a) - i\omega_e C + K_{hys} \Big)^{-1} \tag{3.12}$$

is the RAO or the transfer function which is used to link the input forces with the output response.

3.4 Ship response

The following section focuses on the ship structural response to hydrodynamic loading. Ship response is usually estimated with the help of the finite element method. In this thesis ANSYS finite element software was used for this thesis. The main concentration of this chapter is, however, on the aspects of global strength and ship's global response. The theoretical background of the response analysis is considered from the hydro-structure interaction viewpoint, to make up the whole picture of global strength analysis.

3.4.1 The finite element method

The development of the finite element method (FEM) started in the early 1940s when the complex structural analysis and elasticity problems required a new approach for civil and aerospace industries. Originally, FEM development started from the matrix frame analysis used in the strength of materials. Since then FEM has evolved into one of the most common tools used for structural and fluid flow analyses [3].

The purpose of FEA is to discretize a finite domain with 2D or 3D elements to perform calculations only at a limited number of points. The results at those points are usually interpolated, as can be seen in Figure 3.7.

In principle, the main difference between the FEM and frame analysis is that

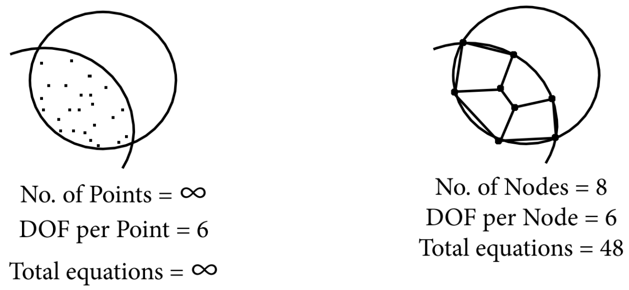


Figure 3.7. Domain discretization using FEM [8]

FEM uses the domain discretization with 2D and 3D elements. This discretization of the domain is called meshing in today's FEA software. However, a discrete model can never be as accurate as of the real continuum. There will always exist an error, arising from the simplification of the real continuum into many finite elements. By increasing the number of elements in the discrete system, the error will decrease, but it will always be present there, no matter how many elements are in the system [3].

When increasing the number of elements, there is a point when the element number will no longer affect the value of the error. By performing the convergence analysis, it is possible to evaluate the point where the element amount increase will no longer have a remarkable effect on the results.

3.4.2 Element types

There are three main types of elements used in finite element analysis of ship structures: beam, shell and solid. Some special elements, which are used to define the boundary condition like springs, gaps and connections were not used in this work and therefore are not covered in this section.

Beam elements are the simplest element types. Long and slender, they are 2D elements which can be located anywhere 3D. In their typical form they may have 6 degrees of freedom at each of 2 nodes, allowing both rotation and translation in every direction [45]. A typical definition of a beam element is presented in Figure 3.8.

As seen in Figure 3.8 the beam element can be defined with node I as a starting point, node J as an endpoint. Node K is used to set the orientation of the beam element's cross-section. In ANSYS beams are based on the Timoshenko beam theory that accounts for rotary inertia and shear deformation effects [9]. However, due to the limitations in the first-order shear deformation theory, beam elements cannot realistically represent shear effects in the transverse direction and are applicable only for slender bodies [9].

Beam elements are usually used for the simplification of such structures as

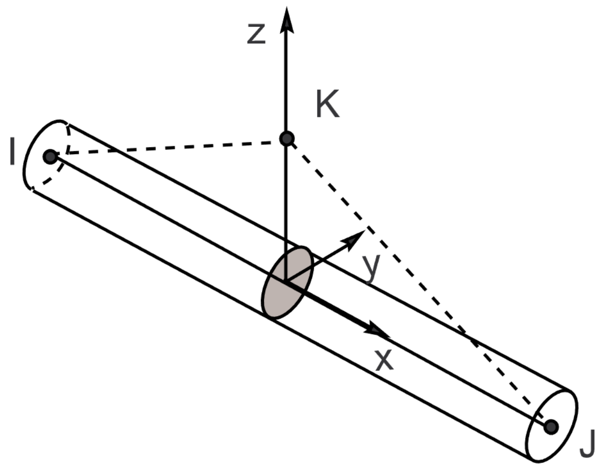


Figure 3.8. Representation of a beam element [9]

stiffeners, girder pillars, and pipes, as this element type can simplify the number of nodes and degrees of freedom in the system and improve the computational time.

Shell elements are at least 3 or 4 node elements with six degrees of freedom at each node. Such elements are representing a 2D plane which can be located anywhere in 3D space. Shell elements are well suitable for analyzing thin-walled structures, where the height to length ratio ($H << L$) is small [9].

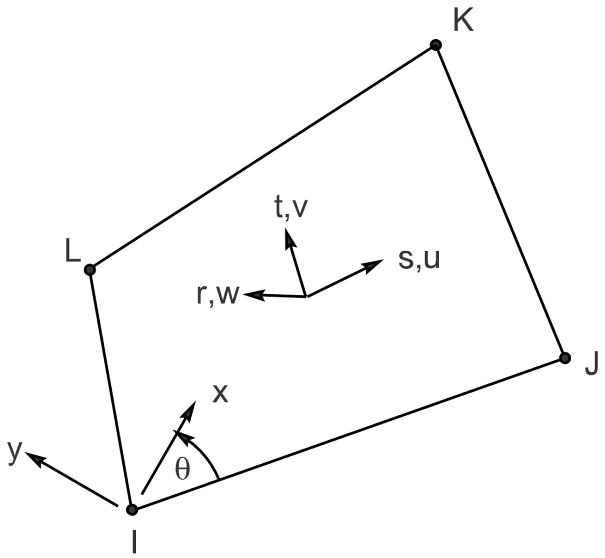


Figure 3.9. Representation of a shell element [9]

Figure 3.9 represents the definition of a shell element in 3D space. Triangular shaped shell elements with 3 nodes are also available. In the area of high-stress concentration triangular elements tend to give worse quality results than quadratic elements [9].

Shell elements are preferred for meshing as they can be meshed in 2D, which is the main benefit of using them. Four nodes of the shell element are used to define the geometry and thickness of the shell must be given additional input data, for accurate physical behavior. Loads can be applied in any direction on a shell, which is a benefit of it in comparison to the beam element. Shell elements are heavily used to model all kinds of structures on a vessel, especially all plate and shell structures.

Last and one of the most complex element types is the solid element. Solid element is 3D element, which has at least four nodes for the pyramid type, and eight nodes for the brick type. The number of degrees of freedom is four for each node. Such elements are used to model solid structures when using other, simpler, element types is not possible.

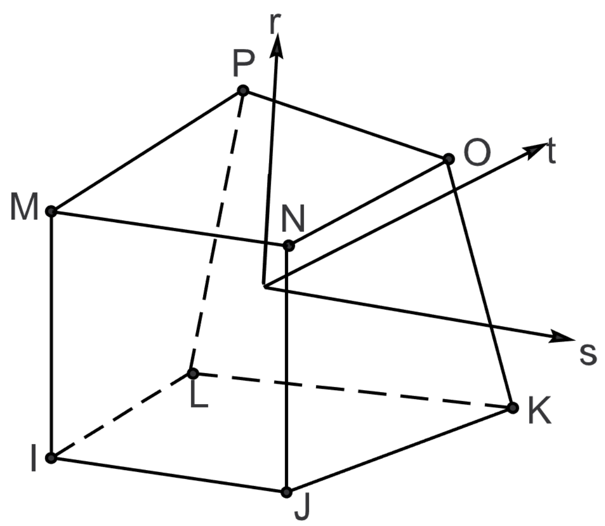


Figure 3.10. Representation of a solid element [9]

Figure 3.10 represents the eight nodes solid element. Solid elements are 3D elements, thus in meshing the internal void of a solid structure is also meshed. This drastically increases the number of nodes and elements in the system.

The correct element type choice is very important in the ship global strength analysis. Model dimensions are large and the number of elements and degrees of freedom are desired to be the least possible. Shell and beam elements are the most common element types when building the global strength model. A simple cantilever beam meshing test can show the effectiveness of each element type. For example, meshing the beam with a beam element of a constant cross-section will result in 12 degrees of freedom. Using shell element with fine mesh will increase this number to about 5500. Solid model will result in 55 000 degrees of freedom. [8]. The total number of degrees of freedom then has a strong influence on the numbers of equations to solve and computational time.

3.4.3 Normal and shear stresses

Various stress patterns are used for the ship response representation. One of the simplest stress representations is normal stress. Normal stress is the stress resulting from the force acting perpendicular to a surface, where the normal stress occurs. Normal stress can be found using the following equation [10]:

$$\sigma_n = \frac{F_n}{A} \tag{3.13}$$

Another type of simple stress occurs when two rigid bodies connected with each other are pulled in opposite directions. The stress happening on the cross-sectional area A between those bodies is called shear stress and can be estimated using the pulling force and the cross-sectional area[10]:

$$\tau = \frac{F_{sh}}{A} \tag{3.14}$$

Figure 3.11 illustrates forces acting on a volume element creating normal and shear stresses. On the right picture axial force perpendicular to the cross-sectional area A will cause the axial stress, which depending on the direction can tensile or compressive stress. On the left picture the shear force, parallel to the cross-sectional area A will lead to shear stress in the control volume [10].

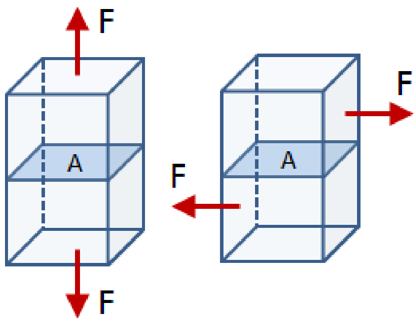


Figure 3.11. Forces creating normal stress (left) and shear stress (right) [10]

Typically, normal stresses can be represented for each direction (x,y,z), or in case of longitudinal strength analysis σ_x is presented. Figure 3.12 represents a typical material under loading with normal stress in each direction and shear stresses in six directions.

3.4.4 Principal stress definition

To judge about the strength of the structure we would need to reduce the number of stresses, so in the end, it would lead to just one stress value. One way to do that is to first use principal stresses.

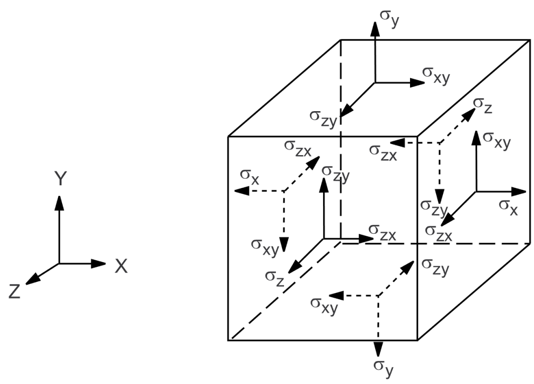


Figure 3.12. Representation of a loaded material volume [9]

The theory of elasticity states that any arbitrary volume of material can be rotated so that there are only normal stresses remain in the body and all the shear stresses are zero. The three normal stresses which remain are defined as maximum, σ_1, middle, σ_2, and minimum, σ_3. These stresses are then perpendicular to the planes of the rotated body, that are called principal planes. The total number of stresses then reduces to three, which allows usage of failure theories such as von Mises yield criterion [9].

3.4.5 Equivalent stress

Von Mises yield criterion suggests that the failure of a ductile material will happen when the von Mises stress, also known as equivalent stress, reaches the material's yield strength. Equivalent stress can be found using the three principal stresses [9]:

$$\sigma_e = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}} \tag{3.15}$$

Equation 3.15 is vastly used in the FEM software and ship design, as it allows any 3D stress state to be described with only one positive stress value. The von Mises stress can be then directly compared to material's yield strength to see if the material can withstand the loading [9].

4. Research

This chapter concentrates on the actual modeling and simulation processes carried out in this thesis. The whole research process is separated into two main parts. The first one considers the FEM model preparation. The second part describes the hydrodynamic model preparation and the load mapping performed between the structural and hydrodynamic models.

4.1 Model ship

To conduct ship global strength analysis a model of a typical middle-size cruise vessel was selected. The ship is a 240 meter long and has eleven decks. The model was previously made using the CADMATIC Hull ship structural software with the quality suitable for basic design. The model of the vessel in solid elements is represented in Figure 4.1. Most of the structural elements were presented in the model, except the smallest local elements like brackets, pillar support plates, etc. HStiffeners and most of the openings were idealized as far as practically possible. Table 4.1 represents the main dimensions of the vessel.

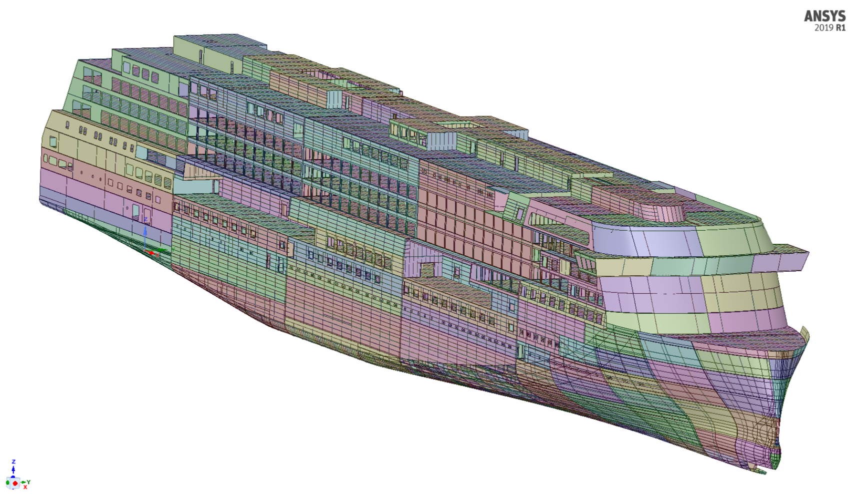


Figure 4.1. Model of the vessel in solid elements in SpaceClaim

Table 4.1. Vessel's main dimensions

Dimension	Quantity	Unit
Length, LOA	240	m
Breadth, B	30.4	m
Draft, T	6.7	m
Displacement, Δ	32500	t

As the model was produced with the basic design quality it is more suitable for the creation of the local structural models. Thus, creating the global model with such quality is a demanding problem as the pre-processing of such a model requires a large amount of time before the analysis. However, such a global model can be a solid base for the creation of the local models, as most of the parts are already transferred in FEM and have an appropriate element type.

4.2 Modelling process

There are various software packages involved in this thesis work, therefore it is important to maintain their functional integrity. As most of the software used comes from different vendors there usually is no straight integration process in terms of file format, etc.

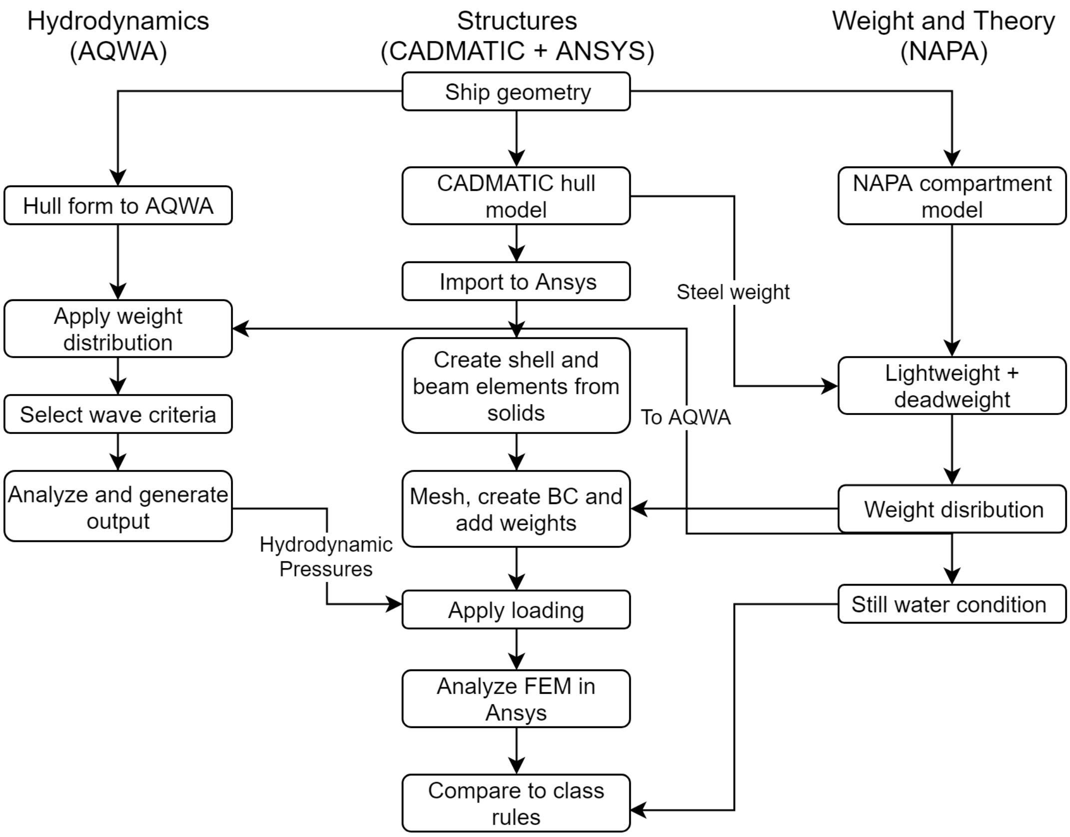


Figure 4.2. Diagram of the process involving the software packages

Figure 4.2 represents the schematic overview of the whole process, where the software integration is described in three different columns. Hydrodynamic calculations were carried out using ANSYS AQWA. Structural calculations were carried out using CADMATIC Hull for modelling and ANSYS Mechanical for FEA. Weight and ship theory calculations were carried out in NAPA.

The modelling process started with the geometry definition and the creation of two important models representing the compartments and key hull steel components. The compartment model was done in NAPA software and the steel model was created in CADMATIC Hull. The ship structural model consists of 6 blocks each having the hull structures and shell plates. After the stuctural model was created, CADMATIC provided the topology for FEA and estimates of steel weight distribution for use in NAPA. Finally, NAPA was used to evaluate the still water bending moment and transfer ship weight data to ANSYS AQWA.

The wave criteria were selected using the wave frequency, wave height and ship's speed. After that, the software was able to produce wave-induced hydrodynamic pressures which were then transferred to the FEM model in ANSYS Mechanical.

Structural model was imported from CADMATIC to ANSYS. Using SpaceClaim, geometry editor tool in ANSYS package, the structural model was optimized and simplified for the further FEA. The FE model was meshed and constrained in ANSYS. Finally, the hydrodynamic pressures were applied and the FE model was solved. Wave bending moment results were compared to the Classification Society guidelines.

4.2.1 Model transfer

The 3D steel model transferring was done using the unified CAD file format — step or stp, as this file format is suitable for SpaceClaim geometry editor. Step file is the most common 3D CAD data exchange format. CADMATIC allows importing 3D model step files in three ways. Firstly, each part's stp file can be created during the 3D model creation in CADMATIC. Another option is to create step file for the selected block or number of parts. Both shell plates and normal parts can be transferred in such a way. Finally, there is an extended step file format, which adds an xml file to the step file created. However, when using the extended step export option the shell plates of the model are not transferred, even if the whole block is selected, which helps to separate the inner structures and shell plates to different files in the importing process.

Due to the large size of the ship model, the transferring process was done in parts, for each block. The shell plates were transferred using the normal step export function, by manually selecting the shell plates in the CADMATIC for each of the blocks. Plates and stiffeners were then transferred via the extended step file format. It excluded the shell plates from the created model and took significantly less time to create such a large step file.

4.2.2 Geometry preprocessing

The 3D model was imported in the SpaceClaim Design Modeller for geometry preprocessing. The main problem of the model imported from CADMATIC was that all the parts come in solid elements. Ship structures are mostly considered thin-walled, where the length to thickness ratio is large. This makes solid elements mostly unnecessary in the finite element analysis, as solid elements have more degrees of freedom and nodes demanding more computational power. Thus, typical element types in the ship structures FE analysis are shell and beam elements. To prepare the model for the analysis, solid elements had to be transformed into shell elements for deck plates, bulkheads and shell plates, and into beam elements for all stiffeners and girders.

Shell and beam elements can be created from solids almost automatically using the SpaceClaim extraction tools. A midsurface can be created for the plate elements, by defining the boundary surfaces of the plate. A software can also automatically create midsurfaces for selected solid elements within a defined thickness range. However, there is an issue in midsurfacing the plates, as most of them are connected to the stiffeners and girders. Extracting a midsurface introduces a gap, equal to half of the plate thickness, between the plate (as shell element) and stiffener, that then requires either a plate or a stiffener to be displaced manually. Figure 4.3 illustrates the discontinuity issue when creating the midsurface from a deck plate. However, when making the midsurfaces from a large amount of parts, for example, the whole ship block, the software is able to automatically connect nearby surfaces and avoid the gaps between the elements.

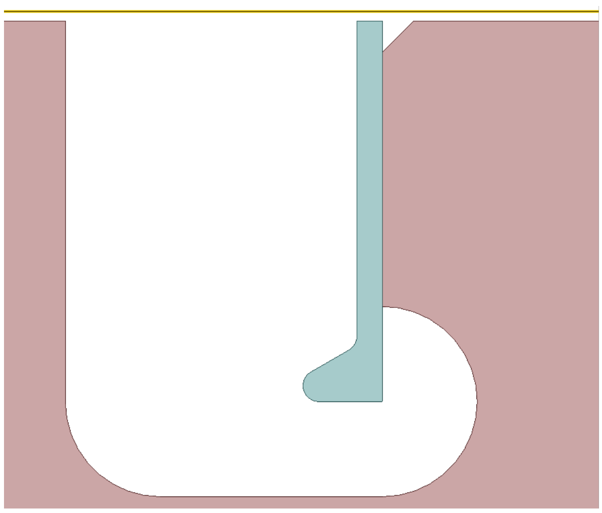


Figure 4.3. A gap between the plate (top), stiffener and girder (both bottom) as a result of midsurfacing the deck plate

When exporting the step model from other CAD software the solid element might not always be exported ideally and might have small holes or extractions. Such imperfections create problems for SpaceClaim when creating using the midsurface function. The solid element is simply skipped when clicking on it with the midsurface tool. No error code comes when the solid is left out. The solution is to use the detach and stitch tools for the solid elements. When using the detach on the solid element it breaks the solid to separate surfaces. When stitching the surfaces SpaceClaim can avoid the tiny imperfections and create a solid, which is possible to midsurface. However, when performing detach and stitch manipulation on the whole block, the software may combine many solids into one, so they will become unusable in further analysis. Thus, the manipulation has to be done in a small portion of parts to avoid this error, for example, one deck at a time.

Beam elements can be also automatically created from solid stiffeners and girders via the beam extraction tool. However, there are a few limitations associated with a model obtained from CADMATIC. Firstly, shell stiffeners have complex curvature and the beam extraction tool needs them to be split in a few, to create a simple beam element. Secondly, the girder creation process in CADMATIC is slightly different from other shipbuilding software, where the girders are created by assigning a cross-section to the desired line. Typically, in ship design software there is a limited number of predefined cross-sections which can be used for beam creation, thus girders are produced by modelling web and flanges as separate plates, or more specifically as plate and face plate.

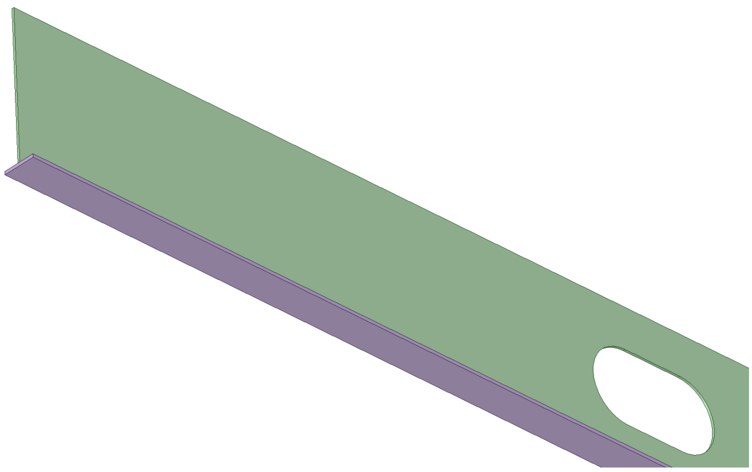


Figure 4.4. Typical T-beam girder consisting of separate web and flange plates

Figure 4.4 represents the typical girder created in CADMATIC Hull. To save computational power, the girder should be preferably transformed into the beam elements. Web and flange could be merged for the beam extraction via SpaceClaim's combine tool or by simply deleted the flange and modifying the beam's, created from the web plate, cross-section. The resulting beam consists then of web and flange combined in the beam's cross-section. However, combining beams is an extremely laborious job, as there is no automatic selection of the elements to be combined and those have to be manually selected one by one. Then the flange and web, of a T-girder for example, have to be of the same length and shape to be combined. If one end of the web is skewed it will not be joined with the flange and has to be straightened and shortened.

Another important issue is the beam element representation in the FEA model. ANSYS as a typical FEA software represents beams as lines with the assigned cross-section. To successfully connect beam elements with shells there are two different conditions. The first one is suitable for pillars and defines that both ends of the beam element have to be lying on the surface of a shell element. As most of the pillars are already connected to the deck and girder in the solid model, this is quite easy to keep. The second condition states that the line of the beam element should lie in one surface with the shell element. Thus, the beam elements have to be moved after they are created so the beam line would lie in the surface of the shell. Then the cross-section of the beam has to be offset so the cross-section would lie right under the shell element, representing the original solid beam. By default, the beam line lies in the area centroid of the beam cross-section and using the location option the beam anchor can be offset to the desired location. There is an option not to transfer beam cross-section when transferring the beam line, which may drastically accelerate the creation of the FEA model.

Although, SpaceClaim has a large variety of options for the creation of beam elements, creating the global model of a cruise ship using beam elements for stiffeners and girders would take a great amount of time. Firstly, the manual combination of girders requires too much time selecting and modifying the solids. Then, beam offsetting should be done separately for beams with different cross-sections. Therefore, it was decided to use create the model using primarily shell elements. However, beam elements are much simpler to solve.

Finally, it was chosen to create the FEM model, which does not include any stiffeners or similar small elements. There were only large plates such as decks, bulkheads, double bottom longitudinal floors and shell left in the model. Such a model was much faster to create than the one with the beam elements and was much easier to mesh that the whole model with only shell elements. Figure 4.5 illustrates the final model used for the FEM calculation. The equivalent plate method was used on the shell of the hull. In this method, the stiffeners of the shell were considered by thickening the plate. The final section modulus of the plate corresponds to the total section modulus of plate and stiffeners.

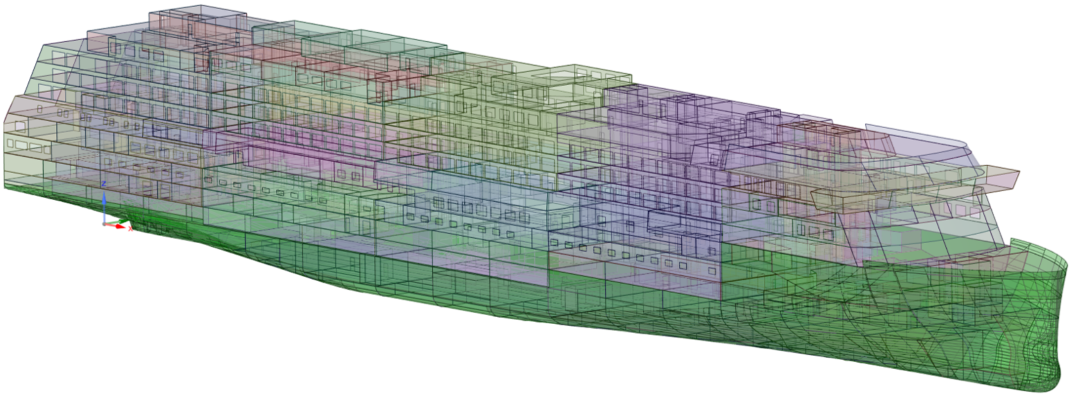


Figure 4.5. The simplified model used for the analyses

4.2.3 FEM model creation

To solve the FEA model the model has to be meshed. For the global finite element model, mesh size is usually quite coarse. Element size must be at least equal to the longitudinal spacing, to separate the deck structure into elements, which are confined by two longitudinal stiffeners. In the rough estimate models, the mesh size can be up to 3 meters. However, class societies give recommendations to have at least one element between both longitudinal and transverse stiffeners and at least three elements between web frames [22].

To properly mesh the model with coarse mesh the minimum element size must be such so that two mesh elements would fit in the part's surface. The size of the smallest part in the model then sets the minimal size of the mesh. Normally, in global models, the girders are the smallest parts modelled using shell element.

Meshing the model properly required simplifications to the structural model. A structural model, which was imported from CADMATIC and midsurfaced in SpaceClaim still contained a large amount of structural details, which would limit the maximum mesh size and software capability to solve the model. Such details can, for example, be small openings and cut-outs, the curvature on the edges of the larger openings and some inexact geometry resulting from the solid midsurfacing. Such small details do not affect the global strength of the vessel, but make the meshing process more complicated. Therefore, it was quired to simplify the model geometry, by removing such small details. When removing those, the total weight of steel in the model decreased, but this could be neglected as the model did not include local structural elements, such as brackets and pillar foundations.

Another important issue done to properly mesh and solve the model is to check the model connections. By default, the model transferred to ANSYS SpaceClaim had parts that did not share topology. This means that the parts were completely free in space do not have any contacts. To create the contacts SpaceClaim has two built-in functions. First, to allow bodies to share the faces, edges, and vertexes, "Shared topology" property must be set to share. However, shared topology is only affecting meshing, allowing to achieve conformal mesh between bodies and meshing the intersection of the bodies perfectly [46].

To connect the bodies in the model, SpaceClaim has an automatic option of first finding the geometry which can share the topology, and then forcing it to share it. There is also an option of specifying the tolerance of the share function, so the bodies located a few millimeters away from each other may be connected. After, using the share function, in the case of the ship model, the software is separating a large surface into the smaller ones at the point of intersection. For example, the deck of the vessel would be split into smaller parts at the point of intersection with the deck stiffeners.

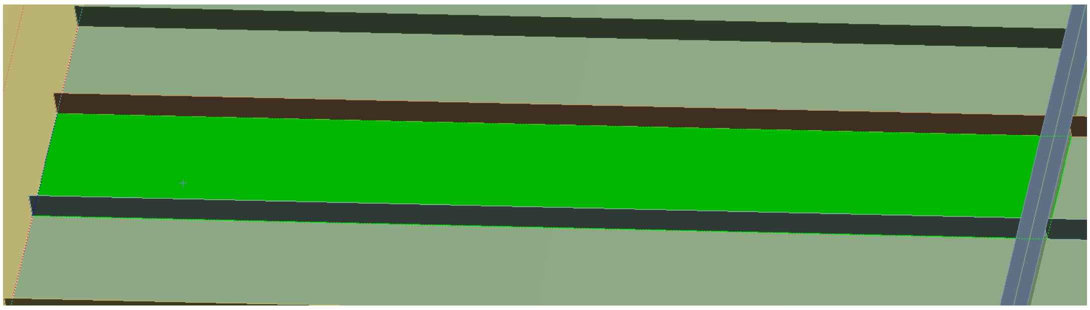


Figure 4.6. An example of the deck element split by enclosing it with other shell elements

The splitting of the large surfaces into small pieces happens only when the split surface is enclosed in the surface by other intersecting shell elements. This situation is represented in Figure 4.6. That is why Classification Societies suggest that the stiffeners should be made using beam elements [47].

To verify that all the parts of the vessel are connected few actions had to be taken. First, the connections were automatically checked using the connections' color code in the shared topology and then extended using the respective tool. The next step involved manually going over the whole vessel model and make sure that large parts are fixed

Another way to check the connections is to run the modal analysis. If parts in the model are free to move they are going to have zero natural frequency. Those parts can be identified and fixed using SpaceClaim.

In this thesis mostly automatic meshing technique was used. Since the small elements had been removed from the model, the meshing process was quite straightforward. ANSYS allows both automatic meshing with the desired mesh sizing and semi-automatic mesh refinements for single geometry objects or model items.

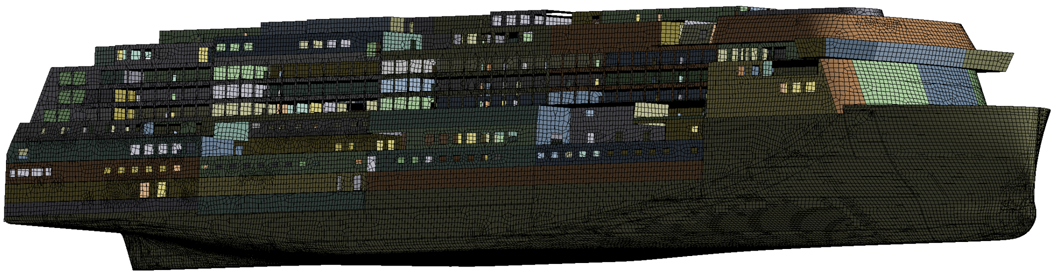


Figure 4.7. The meshing of the whole ship model

Figure 4.7 represents the final mesh of the ship model. In this case, the mesh size was set to be equal to the longitudinal stiffener spacing - 750 mm. The mesh of 750 mm x 750 mm resulted in about 265 000 elements and 252 000 nodes.

4.2.4 Boundary conditions

The FEM model was constrained so there could be no rigid body movement. However, constraints should be properly applied so the model is not over-constrained and reaction forces are not imbalanced.

Class societies provide guidance on the appropriate boundary constraints for the whole ship model. Figure 4.8 represents the constraints according to the Lloyd's Register's Shipright procedure [11]. The same constraints are used among different class societies, for example, DNV GL [48].

The model can be free of the imposed constraints, but then free-body constraints must be used. The model can be then fixed with the reference point, such as the center of gravity. It was decided to stick with the usual constraints used in ANSYS Mechanical, that allow constraining of translation or rotation motions.

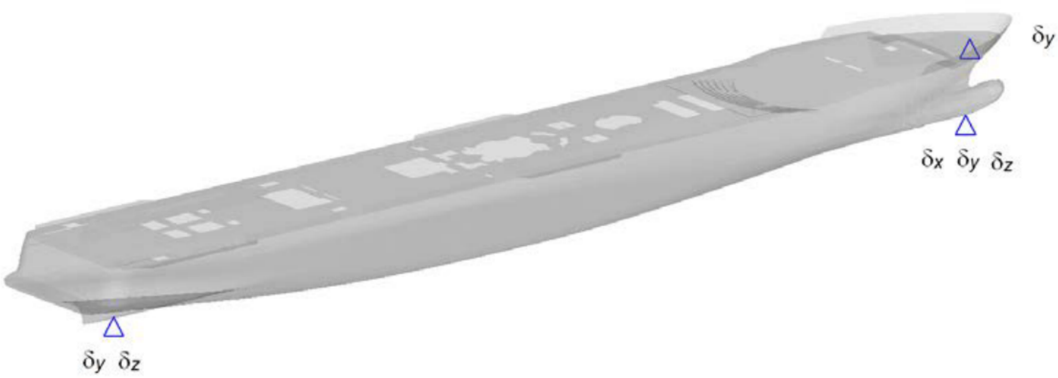


Figure 4.8. Boundary conditions of the whole ship model [11]

The concept of the Classification Society guidance includes three constraints added to the model, according to Figure 4.8:

- At the aft perpendicular on the centerline: translations $\delta_y = \delta_z = 0$
- At the front perpendicular on the centerline: translations $\delta_x = \delta_y = \delta_z = 0$
- At the main deck on the centerline at the front perpendicular: translation $\delta_y = 0$

4.3 Hydrodynamic modelling

In the following chapters, the hydrodynamic application of this thesis work is described. ANSYS AQWA software was used for hydrodynamic analysis. Coupling process with the FEM software is also described as the hydrodynamic pressures were mapped on the hull surface of the FEM model.

4.3.1 Model requirements

As a starting point for hydrodynamic modelling, Ansys AQWA required some initial data. NAPA hull surface was imported as IGES file. It comes as half of the hull and AQWA was not capable of involving symmetry, so later it has to be mirrored and combined to one surface in any kind of geometry editor.

AQWA software is sensitive to the quality of the imported hull surface. The hull surface obtained from NAPA in IGES format contained a lot of small surfaces that were combined to a larger surface to optimize the future meshing process. SpaceClaim could fix the imperfections of the imported surface such as holes and overlapping edges. However, AQWA does not have full functional support of models from SpaceClaim, since it was first saved as a step file once again and uploaded to the ANSYS Design Modeller, another CAD software of ANSYS. Finally, a surface from Design Modeller was used in the ANSYS AQWA.

As Ansys AQWA carries out linear hydrodynamic analysis, only the wetted surfaced of the vessel was of interest. In this process, the user has to make sure that the surface normals of the hull are pointing outwards [12].

Weight items are added as point mass, where the masses of items can be defined using the mass moment of inertia or radii of gyration. By default when the hull surface is added to AQWA it has no structural mass and it must be calculated. With the program-controlled option, the software is able to calculate the displacement of the vessel on the specified trim. However, the software requires the user to input the Z-coordinate of the weight, as well as moments of inertia I_{xx}, I_{yy}, and I_{zz}. Another option is to put the respective radii of gyration in the same direction

[12]. The mass moment of inertia in the x-direction is:

$$I_{xx} = K_{xx}^2 \cdot \Delta/g \tag{4.1}$$

where Δ/g is the ship's mass; K_{xx} is the radius of gyration, that can be estimated based on the vessels main dimensions as [49]:

$$K_{xx}^2 = \frac{(B \cdot C_W)^2}{11.4 \cdot C_B} + \frac{H_e^2}{12} \tag{4.2}$$

In equation 4.2 B is ship's breadth, C_w is the waterplane coefficient and C_b is the block coefficient. H_e is the effective height of the vessel, which can be found by using equation 4.3.

$$H_e = H + V_d/A_d \tag{4.3}$$

where H is the height from the keel to the main deck, V_d is the volume of the superstructure and A_d is the main deck area.

However, the moments of inertia and radii of gyration are available to obtain using the NAPA radii of gyration manager. The manager uses the lightweight table and loading conditions to estimate the moment of inertia and radii of gyration.

4.3.2 Meshing the hydrodynamic model

When the hull surface is prepared, it can be loaded to the Ansys AQWA Hydrodynamic Diffraction module. There the surface can be meshed and the weight of the vessel can be added. The hydrodynamic mesh was generated along the whole hull. In ANSYS AQWA meshing process has certain limitations. Firstly, the meshing can only be controlled with the defeaturing tolerance, or how small details of the model are recognized in the mesh, and maximum element size [12].

AQWA has a limit to the total number of elements and the total number of diffracting elements. The solver is set to have a maximum of 40 000 elements in total of which only 30 000 may be diffracting [12]. Some of the elements in the model can be added as non-diffracting for weight purposes. Thus in comparison to the FEM mesh, the hydrodynamic one is much coarser. To ensure that the maximum number of element criteria is followed, the mesh size of the cruise ship model was set to be about 2 meters. Figure 4.9 represents the meshed hull model with a typical mesh sizing. The total number of elements, in this case, is about 12 000. In the final analysis, a finer mesh was used. The amount of elements was increased, but as a drawback the size of the bow elements enlarged. This resulted in defeaturing of some elements in the bow area. In the case of the overall global strength analysis, this mesh defeaturing does not affect the overall hydrodynamic pressure pattern.

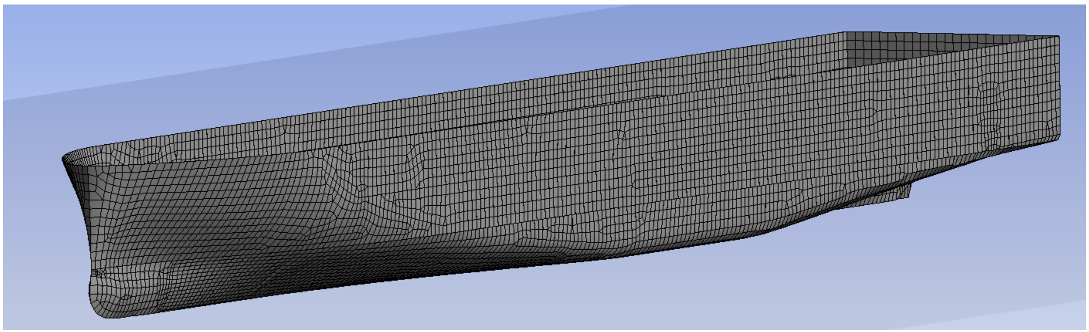


Figure 4.9. Typical mesh sizing of maximum 2 meters used in the hydrodynamic analysis

Mesh size parameters depend on maximum wave frequency which is used in diffraction analysis. This limits the maximum wave frequency used in the system, as the total amount of elements is also limited in the software. Thus the model must be meshed based on the maximum allowed amount of elements and smaller mesh size, which will lead to the maximum wave frequency. Wave frequency is directly related to the maximum mesh size and there is an option to set the desired maximum frequency value instead of the mesh size. The smallest allowed element size is dependent on the water depth [12].

4.3.3 Wave parameters

There are various wave parameters in ANSYS AQWA that define the diffraction/radiation analysis. First of all, there are two kinds of analysis type namely: "hydrodynamic diffraction" and "hydrodynamic response". The main difference is that the "hydrodynamic diffraction" analysis is performed in the frequency domain and "hydrodynamic response" uses a time domain approach. However, the mapping of hydrodynamic pressures is only possible using the frequency domain analysis, since the "hydrodynamic diffraction" analysis was chosen [12].

One of the parameters used in the "Hydrodynamic diffraction" is the wave direction, which also includes the forward speed parameter. There are various types of defining the direction. Forward speed can have either zero value or non-zero constant value in one single analysis system. There is no option in ANSYS AQWA to run the analysis with increasing forward speed. When a range of directions is selected, waves can be applied from -180 to +180 degrees on a specified interval [12].

Another important wave parameter is the wave frequency. As already mentioned in Section 4.3.2 values of maximum and minimum wave frequencies depend on the mesh size and water depth specified for the analysis. The maximum wave frequency is set by the mesh size. The value of the lowest wave frequency in rad/s is estimated using the following equation:

$$\omega_{lowest} = 0.001 \cdot \sqrt{\frac{g}{d}} \tag{4.4}$$

Where g is the gravitational acceleration and d is the water depth. However, the lowest allowed frequency must be at least 0.1 rad/s.

Wave frequency can be defined as a single value for the whole analysis or as a range of frequencies. The range is the default option, allowing to set the desired number of wave frequencies with constant increment.

In hydrodynamic diffraction analysis, AQWA can automatically produce encounter frequency for specified forward speed and single or a range of wave directions.

4.3.4 Hydrodynamic solution

Before conducting any hydrodynamic analysis, AQWA requires to solve the hydrostatics of the model. In the hydrostatic section, the vessel was analyzed using the still water criteria and small angle of stability. Based on the input moments of inertia and Z-coordinate center of gravity, ANSYS AQWA can output the vessel's center of gravity, the center of buoyancy and displacement. Some stability values can be also produced, such as metacentric height, out of balance moments and restoring forces.

The hydrodynamic model is then solved using the panel method. In the approach, a pulsating source is virtually placed in the centroid of each of the panels. The wetted surface is separated into panels. Frequency domain Green's function is introduced to solve the velocity potential on each of the panels. A Green's function database is used to solve the Green's function and its derivatives to save the computational time [50].

Various graphical results were produced using ANSYS AQWA. Response amplitude operators for every motion type, the variation of the exciting forces, share force and bending moment diagrams, added mass, radiation damping plots were made using the software and are presented in the discussion chapter.

Hydrodynamic pressures on the underwater part of the hull can be graphically represented. For a selected frequency and wave direction, wave elevation and pressure distribution were produced.

The user has to consider that ANSYS AQWA has some limitations in the radiation/diffraction analysis. Inputting large wave amplitude and high wave frequency values will result in unrealistically large wave elevation and pressures.

4.3.5 Fluid-structure interaction

Mapping of hydrodynamic loads to the structural model is one of the functions of ANSYS software package. Wave loading can be transferred directly to the FEM model, once the hydrodynamic diffraction analysis is performed. Mapping has been done with the help of the Hydrodynamic Pressure Mapping ACT Extension. However, only frequency domain results can be transferred to the structural model, thus the time domain analysis was not used.

Hydrodynamic pressure mapping tool works as a link between the hydrodynamic diffraction and static structural packages in ANSYS Workbench. The typical workflow using the mapping tool is presented in Figure 4.10.

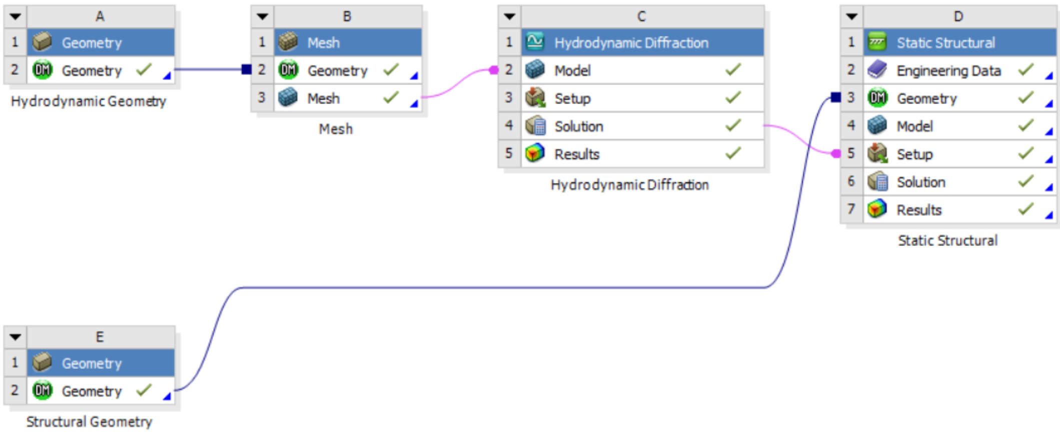


Figure 4.10. An example of a workflow for ANSYS Hydrodynamic pressure mapping tool [12]

The hydrodynamic mapping tool allows the transfer of the linear pressure terms including hydrostatic, incident, diffracted and radiated. Loads can be mapped at a single wave phase angle, over several phase angles making up the whole wave cycle or for real (0°) and imaginary (90°) components. The sum of pressure components is then mapped to the structural mesh [12].

The mapping tool can be configured inside the static structural analysis. Most of the parameters used in the hydrodynamic diffraction are also available in the structural analysis. The identical forward speed used in AQWA may be transferred to the analysis. Wave direction and wave frequency can be selected internally in the static structural analysis, so separate runs of hydrodynamic diffraction were not required when varying the parameters. Incident wave amplitude was also given in the same window.

There are two mapping configurations available using the hydrodynamic pressure mapping tool. First is the interpolated method. By using this method the pressures are interpolated from the center of the hydrodynamic mesh elements onto the nodes of the structural mesh. This method is suitable for models with forward speed but the quality of the interpolated loading depends on the hydrodynamic mesh. The second method is the direct method, where the position of the nodes is estimated by calculating the strength of the diffracting panel source in the hydrodynamic analysis. However, this method is not suitable for setups with forward speed.

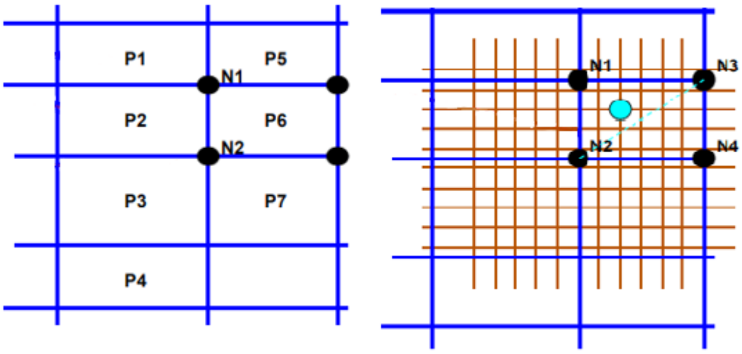


Figure 4.11. Mapping principle [12]

Figure 4.11 represents the interpolating method used via ANSYS AQWA for the purposes of this work. After the hydrodynamic calculation was carried out pressures (P) needed to be mapped onto the structural mesh. First pressures at the nodes (N) of the hydrodynamic mesh were estimated from the panel pressures. Then the node pressures of the hydro mesh were interpolated onto the structural mesh nodes. The pressure at the nodes then was interpolated onto the centroids of the structural mesh elements.

Typically, in the hydrodynamic analysis vessel's coordinate system is displaced in Z-direction as the underwater part of the hull must be under water. To account for this issue the vessel's coordinate axis can be transformed in the static structural analysis.

Figure 4.12 illustrates the hydrodynamic pressures mapped on the structural model. The wave pressure corresponds to the six-meter wave at $0°$ phase angle and speed of 15 knots. Maximum pressure, in this case, acts on the midship part of the vessel which corresponds to the hogging condition.

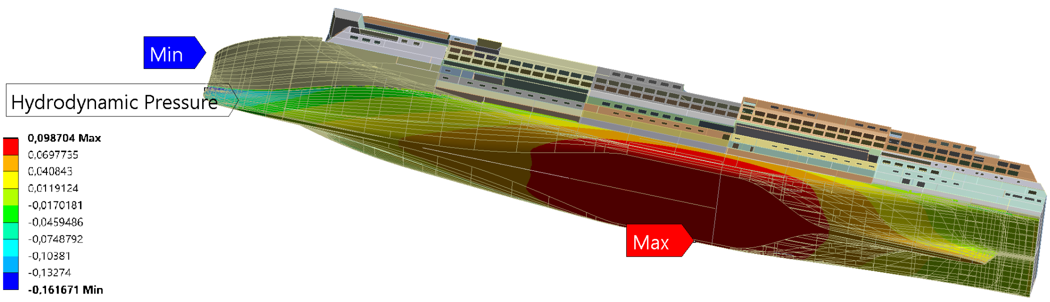


Figure 4.12. Hydrodynamic pressures mapped on the hull surface of the structural model

The pressure represented on the color legend is in MPa and should correspond to the hydrodynamic pressures in ANSYS AQWA. The quality of the mapped pressure highly depends on the balance of the hydro and structural models. This means that the weight and weight distribution of the hydrodynamic model must be the same as used in the structural model. In the hydrodynamic model, the weight is adjusted by applying correct moments of inertia and center of gravity. In the case of structural model, the weight must be adjusted by applying the weight elements as additional mass or area load on decks. In this thesis work, the balance was suppressed.

5. Discussion

This section concentrates on the results and outcomes of the thesis. Results for both hydrodynamic calculation and ship responses are presented and outcomes of the results are discussed. Finally, conclusions and future development of the model are presented.

5.1 Still water condition

Still water condition was executed using NAPA. Figure 5.1 represents the still water graphs obtained from the software. Bending moment (BEND), shear force (SHEAR), weight (WD) and buoyancy (BD) distribution are represented.

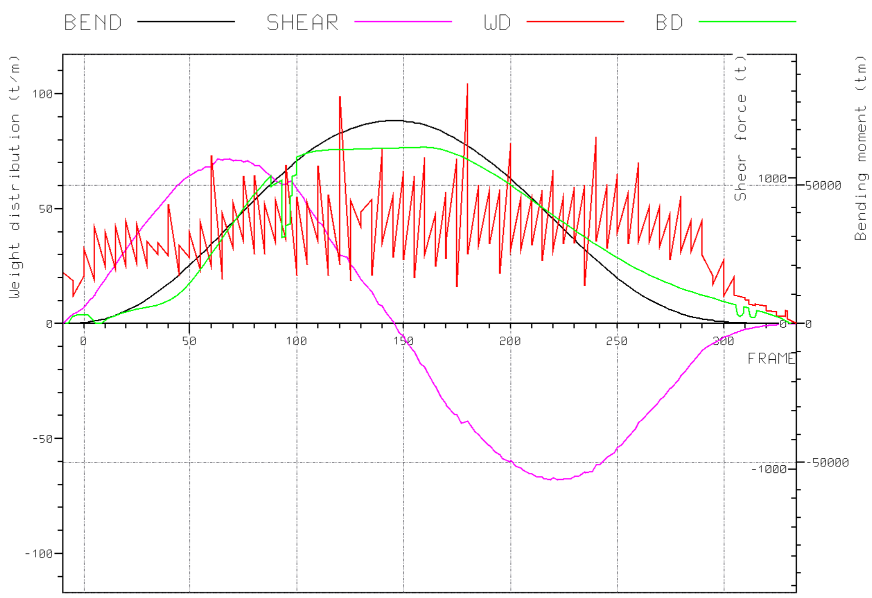


Figure 5.1. Still water condition graphs

Bending moment and shear force diagrams represent typical distribution with bending moment maximum at midship and shear force maximums at some distance from aft and fore parts of the vessel. On the graph weight distribution has sharp peaks because of the steel weight transferred from CADMATIC Hull. The steel weight in CADMATIC is given as a weight at a point, not on an interval, making the weight distribution graph look sharp. The buoyancy distribution has some missing parts at about frame #80-100. This is due to some missing compartments in the NAPA model and this issue should not affect the still water condition on a whole scale.

The maximum still water bending moment value from the curve is about $72\ 000\ tm$, which is about $706\ 000\ kNm$. The maximum allowed value using the DNV GL class rules is $152\ 0000\ kNm$ which is twice as much as the obtained value.

5.2 Hydrodynamic analysis

The hydrodynamic analysis was performed for several cases. First two vessel speed cases were used - zero and 15 knots. Then ship motions RAO were produced for two headings - head seas ($180°$) and oblique seas ($135°$). This parameter affected ship motions RAO and hydrodynamic pressures. Finally, the hydrodynamic pressure results are represented for two incident wave heights namely: 1 and 6 meters. The most valuable results are represented in this chapter, while others had been added to the appendix.

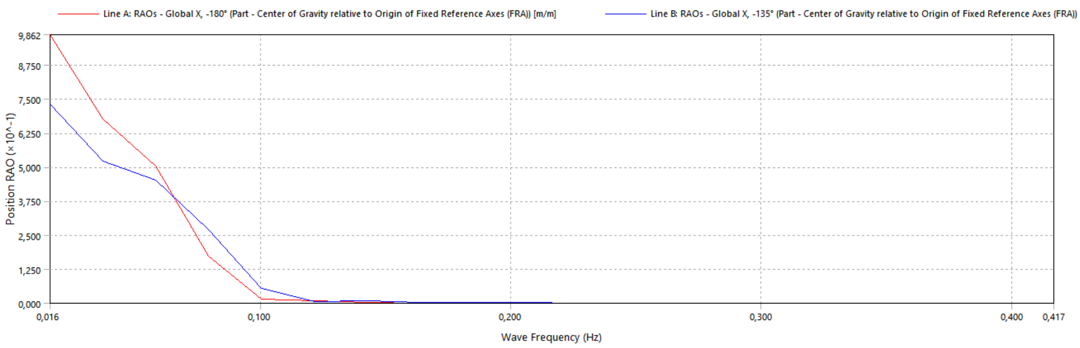
5.2.1 Ship motions RAO

Response amplitude operators of ship motions predict the behavior of the vessel in waves. In the global strength analysis, three motions have the most influence on the longitudinal response. Those are surge, heave and pitch making the vessel to bend longitudinally, while sway, roll and yaw motions are mostly influencing the transversal response of the vessel.

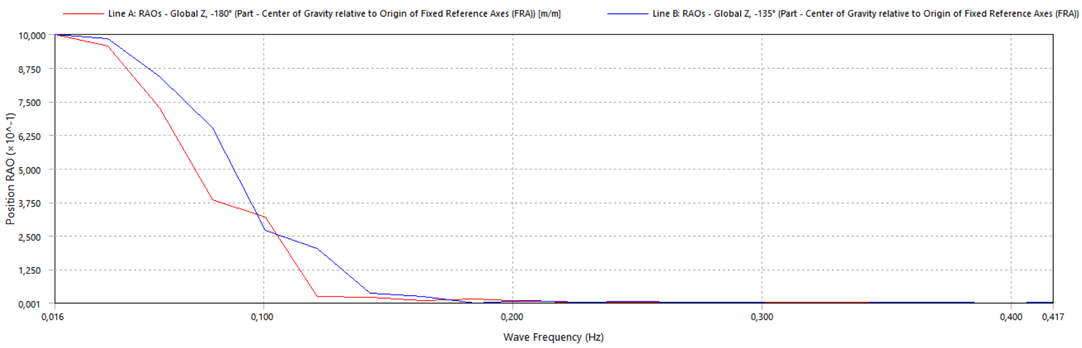
Figure 5.2 represents RAOs for surge, heave and pitch motions for $180°$ and $135°$ at 15 knots. RAOs are represented as functions of wave frequency and position RAO in m/m for the translation motions or rotation RAO in $°/m$ for rotational motions.

As can be observed from Figure 5.2 RAOs of the presented motions are mostly distributed at the low frequency range from 0 to $0.15Hz$, which corresponds to a longer wave. Overall, the shape of the RAOs follows the typical RAO from for surge, heave and pitch. Surge motion RAOs represented on Figure 5.2a have a similar shape for both $180°$ and $135°$ headings. The $180°$ heading RAO has a higher value at the low frequency, meaning that the surge motion will be more active when the wave is longer than the length of the vessel. After the corresponding frequency, the $135°$ heading starts will have a higher RAO value.

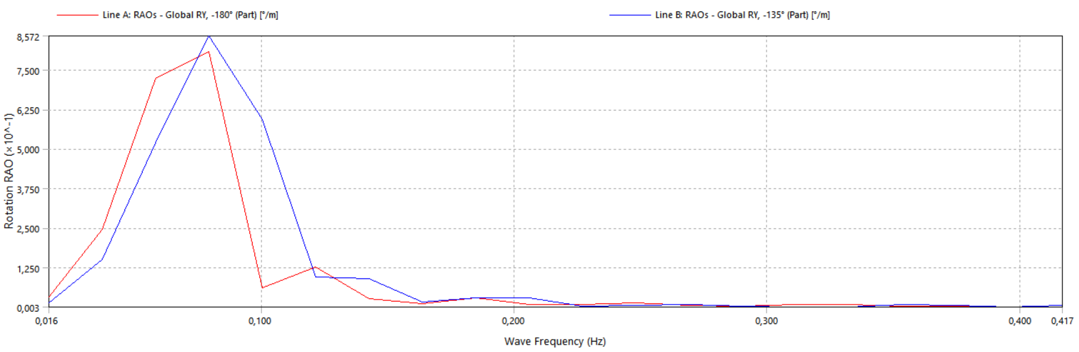
The heave RAOs represented on Figure 5.2b also have a very similar pattern for both headings, however, the $135°$ heading RAO has a higher value almost along the whole frequency range. Pitch RAOs demonstrated for both heading on Figure 5.2c are both reaching the highest values at $0.08\ Hz$ a value that corresponds to the wave length equal to the ship's length. This will result in maximum heave motions for both headings when the vessel encounters this wave length. The overall shape of the pitch RAOs is similar, but the $135°$ RAO is shifted towards the higher frequency range.



(a) Surge RAO



(b) Heave RAO



(c) Pitch RAO

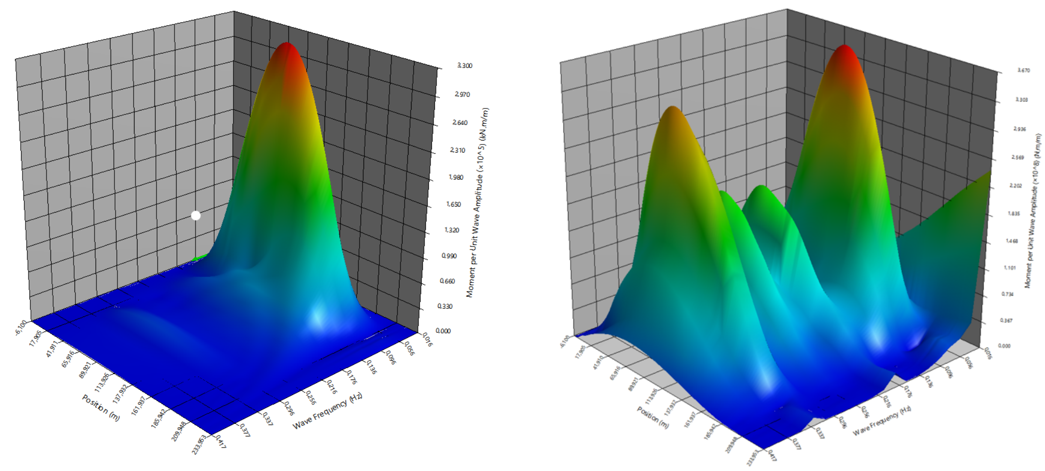
Figure 5.2. RAOs at $180°$ and $135°$ headings at 15 knots

Surge, heave and pitch RAOs for zero forward speed case are represented in Appendix A.1. The overall shape of RAOs is quite similar, as in the forward speed case, however, values of the zero speed case are lower.

RAOs of sway, roll and yaw are represented in Appendix A.2. Values of the functions in oblique seas are much higher than the head wave case. It is thought that this occurs because the ship does not experience any transversal disturbance in head waves. Thus RAOs for these motions are represented on separate plots.

5.2.2 Bending moment and shear force RAO

ANSYS AQWA can represent response amplitude operators for bending moment and shear forces as a 3D function of few variables. Figure 5.3 represents bending moment RAOs for zero and 15 knot forward speed cases. The right axis is the wave frequency, the left axis is the position along the ship's length and the vertical axis represents RAO in terms of moment per unit wave amplitude.

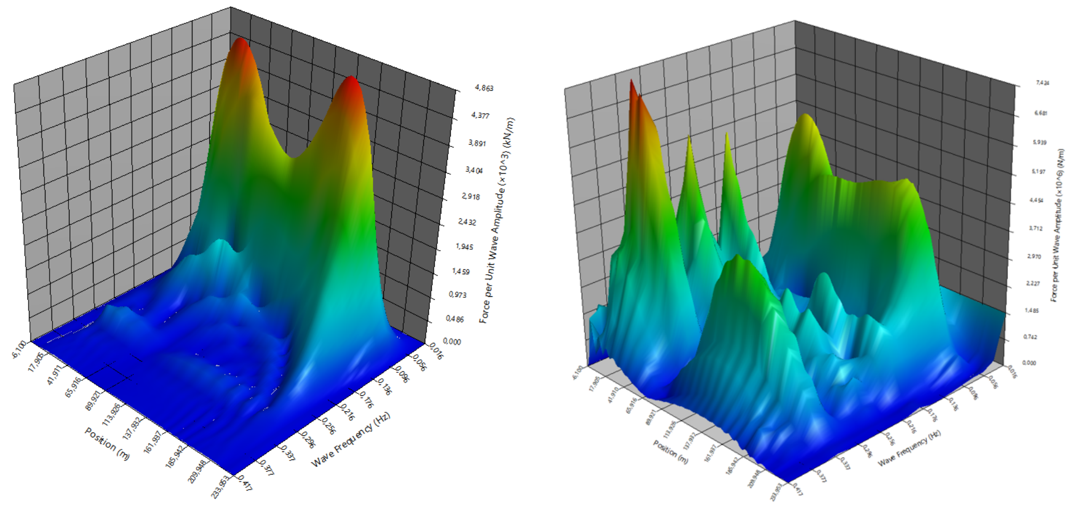


(a) Zero speed case **(b)** 15 kn forward speed case

Figure 5.3. Bending moment RAO

Figure 5.3a represents the bending moment RAO for the zero speed case. There is only one high peak, where the bending moment is the most active. This happens at the frequency range which corresponds to the wave length equal to the ship's length. This occurs as the ship stiffness is able to absorb the wave exerted forces at any other frequency. The highest point of the bending moment RAO happens amidships.

The second plot, Figure 5.3b, illustrates the forward speed case. As we can see from the plot, the ship stiffness is no longer able to absorb the motions on cases different from the wave length equal to the ship's length. In this case, there are two major peaks on the plot. Position of the first peak corresponds to the zero speed case, but the maximum value is slightly higher in the forward speed case. Once again, the maximum bending moment will occur amidships at the wave frequency of about $0.08\ Hz$. However, there is a second major peak which happens in the high frequency range. The maximum value of this peak is lower than the first peak's. Since locally the value of bending moment may be higher, thus in order to see the whole picture on the vertical wave bending moment, two peaks have to be combined. In the intermediate frequency range between the two highest peaks, the value of bending moment RAO is moderate and is about two times lower than the bending moment RAO of the lower and higher frequencies.



(a) Zero speed case

(b) 15 kn forward speed case

Figure 5.4. Shear force RAO

On Figure 5.4 the shear force RAO is represented. The axes are identical to the bending moment RAO, except that the vertical axis now represents force per unit wave amplitude. For zero speed case has the shear force is active only in the low frequency area. The maximum value consists of two peaks which form symmetrical distribution of the shear force along the vessel's length. At other frequencies, the vessel is not prone to the shear force acting on it, due to lower motions in the zero speed case.

For the forward speed case the highest values of shear force acting on the frequency, where the wave length is equal to the ship's length, and on the higher frequency range, where the highest peak can be observed (see Figure 5.4b). However, the highest values of the shear force are about 40% higher in the forward speed case than in the zero speed case.

5.2.3 Wave bending moment

Classification Societies provide Rules and Guidelines on longitudinal strength. These Rules and Guidelines provide limitations on maximum allowable wave bending moment. The maximum value is based on ship statistics and empirical data. In order to compare the class values for the wave bending moment with the results, the wave bending moment diagram was produced.

The bending moment RAO represented in Figure 5.3b was combined into one plot, using the two highest peaks. To do this, the values from two peaks at the frequency of $0.08\ Hz$ and $0.35\ Hz$ were collected and then the maximum value from each of the peaks was taken. Then using the incident wave amplitude of six meters the values were plotted as a function of the ship's length and wave bending moment and are represented in Figure 5.5 as a blue line. The wave bending moment limit value according to DNV GL class rules is plotted in red [51].

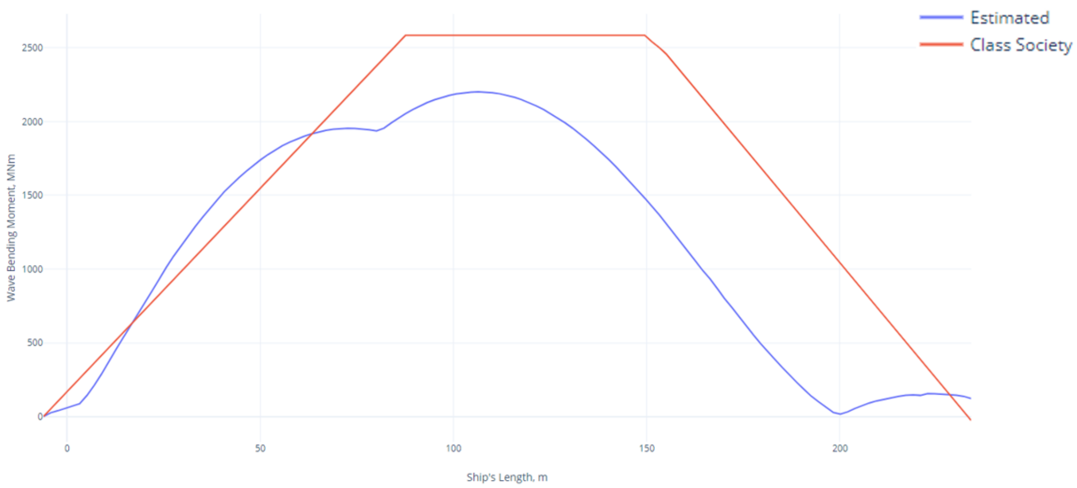


Figure 5.5. Wave bending moment comparison

As can be seen from the estimated value plot there are two local extremes from the peak values of the bending moment RAO. The bending moment values from 0-70 meters are the values based RAO at $0.35\ Hz$. The rest of the values from 70 meters are taken at $0.08\ Hz$. The whole graph is shifted to the left side, which happens possibly because of discrepancy between longitudinal center of gravity and center of flotation considered during hydrodynamic modelling.

In the fore part of the resulted wave bending moment, there is a slight leap, which can be associated with the bow form of the vessel. Moving from amidships the wave bending moment decreases, but then there is a small increase. This may be due to the fact that the cross-sectional area at the bow of the vessel is quite small. Additionally, there are small mesh defects, that could cause the moment to first decrease to the lowest value in the fore part.

When comparing the resulted wave bending moment with the Class Society limitation value, there are few interesting points. Firstly, in the midship area, the Classification Society value ($2580\ MNm$) is way higher than the maximum value obtained in the simulation ($2200\ MNm$). This may happen due to the fact that Class Society might overstate the midship value for safety reasons, as a failure is most likely to happen amidships.

Another point is that the resulted wave bending moment has a higher value in the aft part of the vessel. This may be caused by a number of issues. The first issue may be related to the Classification Society rules being too concentrated on the midship area, when the aft and fore part of the vessel may require more attention. The maximum value of the wave bending moment is considered to be extended over 30 % of the length of the vessel in the class rules [51]. However, this might only mean that the Classification Society uses an approach which is different from the approach used in this thesis. Thus, the validity of the approach used in this thesis requires future investigations. Another issue might be the definition of the center of gravity in AQWA model which probably caused shifting of the wave bending moment graph.

5.2.4 Added mass and radiation damping

Figures 5.6-5.7 illustrate the distribution of the added mass and radiation damping along the encounter frequency domain. The calculations were done using 15 kn forward speed case. Graphs for zero speed case are available in Appendix B.

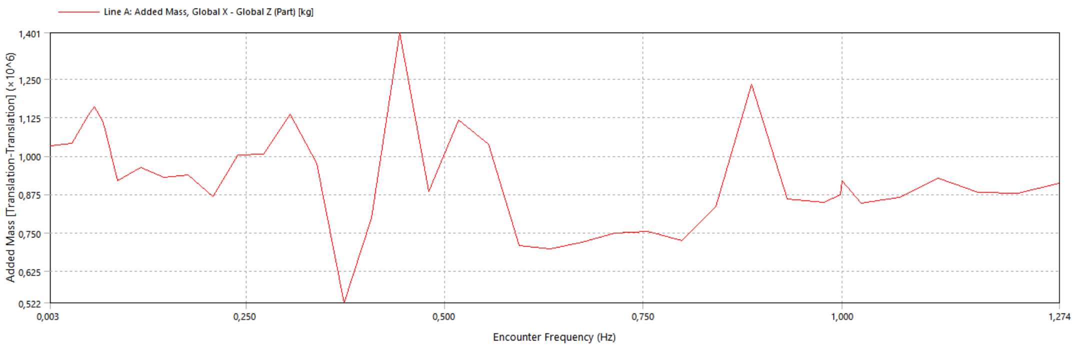


Figure 5.6. Added mass distribution at 15 knots

Added mass distribution on Figure 5.6 shows the added mass in kg along the encounter frequency domain. The average value for the whole domain is about 900 ton, which corresponds to about 3 % of the total ship mass. The maximum value of the added mass traveling with the vessel is 1400 ton and happens at the encounter frequency of $0.44\ Hz$, which is about 4.3 % of the total ship weight. In the minimum value, the added mass of 521 ton happens at $0.376\ Hz$.

Radiation damping distribution is presented in Figure 5.7 as a function of $kN/(m/s)$. Along the whole encounter frequency domain, the radiation damping value is low and negative at some points. However, at the encounter frequency of $0.376\ Hz$ value of the radiation damping rises drastically. This event happens at a similar frequency to the minimum added mass traveling with the vessel on Figure 5.6. It is possible, that in the event of the minimum added mass the radiation damping has to compensate for the total radiation force component.

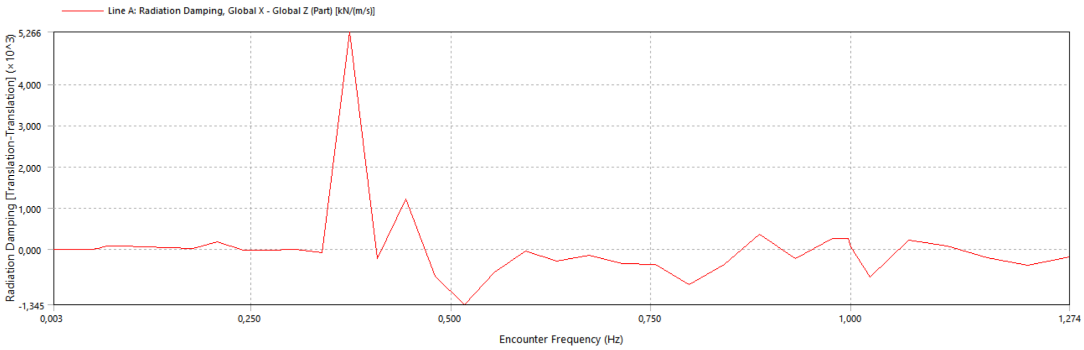


Figure 5.7. Radiation damping distribution at 15 knots

5.2.5 Hydrodynamic pressures

Before the structural analysis could be executed the hydrodynamic wave-induced pressures had to be produced. Calculations for the hydrodynamic pressures were performed for several parameter cases:

- Forward speed of 15 kn and zero speed case
- Incident wave amplitudes 1 m and 6 m
- Headings for head waves (180°) and oblique waves (135°)
- Wave frequency of $0.08\ Hz$, which corresponds to $\lambda = L_{OA}$

In total there were 8 result sets produced. For strength analysis, the highest pressure acting on the ship's hull arose the most interest. This case corresponds to the 15 kn forward speed, 6 m incident wave amplitude and both headings.

The wave frequency of $0.08\ Hz$ was selected as its wave length is then equal to the length of the vessel. Also, ANSYS AQWA has limitations on the panel code so cases with the forward speed and high frequency will produce irrelevant results. Headings of $(180°)$ and $(135°)$ were chosen for the analyses as they were believed to have a higher influence on the longitudinal strength of the vessel.

Both heading cases were executed at the $0°$ wave phase angle. Such a phase angle is used to simulate hogging condition. When simulating in hogging condition the high pressure area was considered to be the largest and thus have a higher influence on the global strength of the vessel. Simulating sagging condition using AQWA turned up to be quite challenging. As the vessel is rigid, it was following the wave, so the bow or the stern of the vessel was eventually in the water. This behavior of the vessel in waves interfered simulating sagging condition.

Figure 5.8 represents results for the head wave of six meters at 15 knots. The highest positive pressure is in this case acting amidships, which usually corresponds to hogging condition. The maximum pressure acts on the sides of the hull

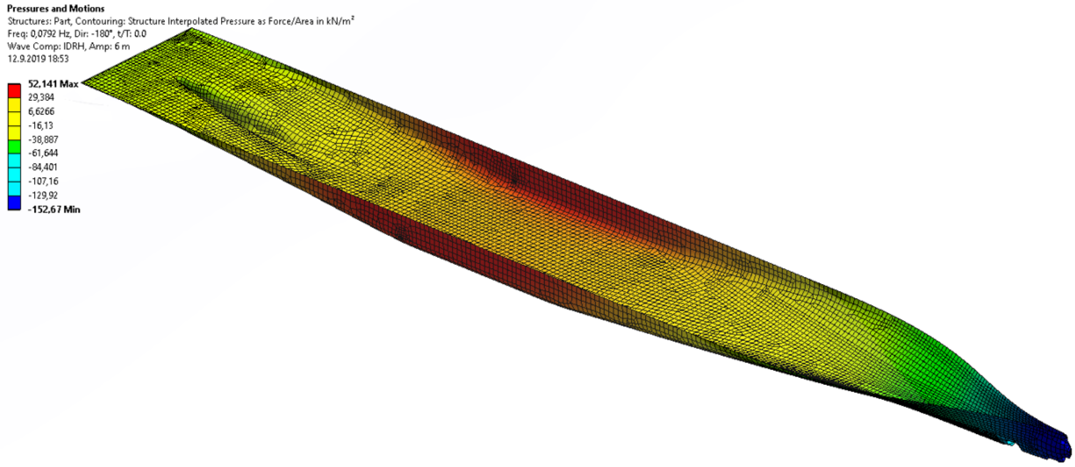


Figure 5.8. Hydrodynamic pressures of 6 m head wave ($180°$) at 15 knots

and was about 52 kPa. However, there is a high negative pressure acting on the bow of the vessel, which is 3 times higher than the maximum positive pressure in the absolute. Such large negative pressure is due to the ship's bow being in the air, which is also typical in hogging. Similarly, the aft also experiences high negative pressure.

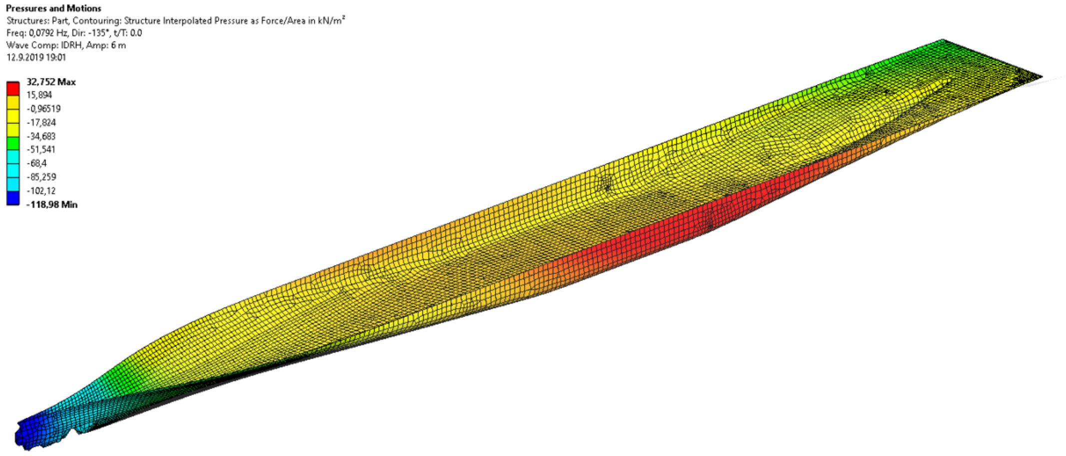


Figure 5.9. Hydrodynamic pressures of 6 m oblique wave ($135°$) at 15 knots

Figure 5.9 illustrates the second heading for the oblique seas. In this case maximum positive and negative pressures are lower than in the head seas. Maximum positive pressure this time acts only of the portside of the vessel. As a wave is approaching the vessel at $45°$ from portside, the pressure is higher there and covers a larger area, than in the starboard. The negative pressure similarly to head waves has a higher value than the positive pressure and acts in the bow and stern areas.

To compare all cases, results were combined into two tables for positive and negative pressure. Table 5.1 represents positive pressure values. The highest values correspond to the forward speed 6 m wave amplitude case in head seas. Worth noting, that in case of oblique seas the value in zero speed is greater than with forward speed. This may be related to the vessel being more prone to roll motions in zero speed. So when the vessel rolls its portside has higher acceleration when contacting with the water surface, which leads to higher pressure in zero speed case.

Table 5.1. Maximum values of positive pressure for all cases

	$0\ kn$		$15\ kn$	
	$180°$	$135°$	$180°$	$135°$
$A_w = 1\ m$	$6\ kPa$	$6\ kPa$	$8.7\ kPa$	$5.5\ kPa$
$A_w = 6\ m$	$39\ kPa$	$37\ kPa$	$52\ kPa$	$33\ kPa$

Table 5.2 presents the maximum negative pressure values in absolute. The highest values also correspond to forward speed case. When comparing both tables there are two notable trends. Firstly, in the case of zero speed, values for both headings are almost equal. Though the waves act differently, there are different ship motions, which result in the same values for pressure. Secondly, in the zero speed case, the values of negative pressure are about two times higher in absolute than the positive pressures. However, in forward speed, the negative values are 3 times higher than the positive ones. This may be the result of the larger bow extent being in the air when the ship has forward speed. This difference may also depend on the forward speed effect considered in ANSYS AQWA.

Table 5.2. Maximum absolute values of negative pressure for all cases

	$0\ kn$		$15\ kn$	
	$180°$	$135°$	$180°$	$135°$
$A_w = 1\ m$	$12\ kPa$	$12.6\ kPa$	$25\ kPa$	$22\ kPa$
$A_w = 6\ m$	$76\ kPa$	$76\ kPa$	$152\ kPa$	$119\ kPa$

5.3 Ship response

Hydrodynamic pressures were mapped on the structural model using ANSYS Mechanical solver. Results were produced for normal stress in x-direction and equivalent stress along the vessel. Average equivalent stress values were also produced for three frames amidships.

In structural analysis only pressure at 15 knots in head waves and 6 meters forward speed was chosen as being the most representative.

To ideally transfer the hydrodynamic pressures, both structural and hydrodynamic models have to be balanced. Due to removed stiffening, in the structural model the weight balance of the models was disregarded in this thesis. The structural model in its current state does not represent reality but it may be sufficiently indicative for global strength estimation.

There were large deformations in the deck structures during the FEM analysis. Structure lacked stiffness and could bend under its weight. However, in a larger scope, it did not affect the global strength analysis procedure.

Due to high stress values in decks it was difficult to judge about the critical stress. However, stress results allowed visualizing the influence of the wave-induced pressures mapped on the hull and the overall ship response.

5.3.1 Normal stress

Figure 5.10 represents the normal stress in the x-direction, σ_x, distributed along the vessel. There are both large positive and negative stress values in the model. The largest values of stress occurring in the deck structures with large deformations were disregarded, to represent only valuable results. Limiting values of 60 and -60 MPa were chosen for the normal stress distribution on the color code legend. However, the legend takes into the account stresses that are larger than 60 and -60 MPa.

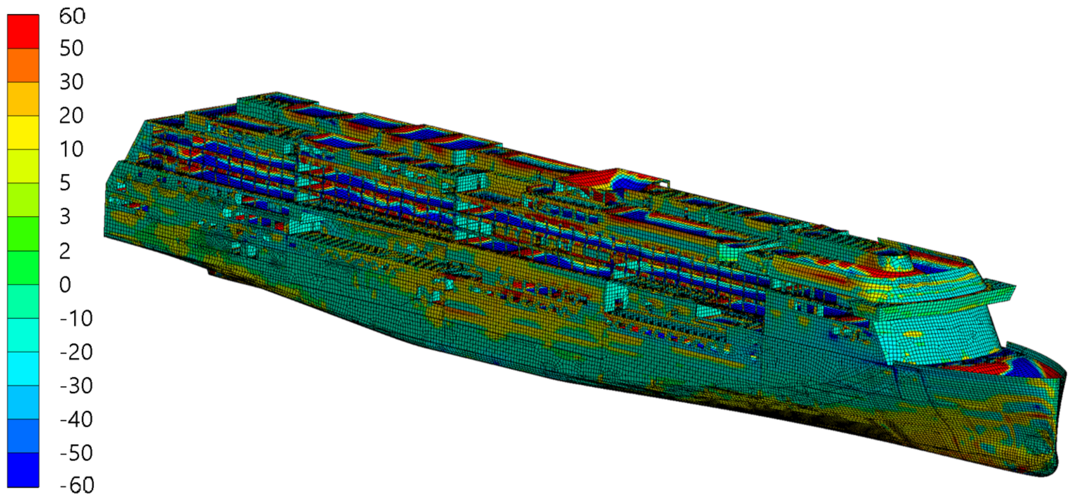


Figure 5.10. Normal stress σ_x distribution along the vessel in MPa

In way of the wetted surface of the ship's hull there are positive normal stress values of $5 - 50$ MPa. Stress variations may also be seen in the bow area, where the negative pressure acts along with the weight of the bow. Most common stress in the model is in the range from 0 to -10 MPa.

5.3.2 Equivalent stress

Figure 5.11 illustrates the equivalent (Von Mises) stress distributions. Similarly to the normal stress, there were large stress values in the deck structures, caused by not stiffened plate deformations. Such deformations typically happen in the decks with large span, which are not supported by pillars and bulkheads.

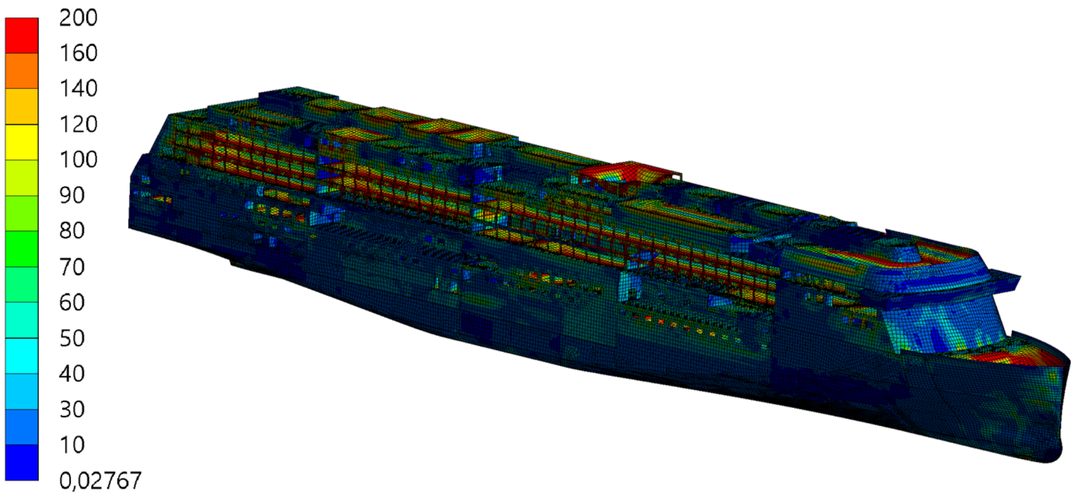


Figure 5.11. Equivalent stress σ_e distribution along the vessel in MPa

The values of equivalent stresses observed range from 10 to 80 MPa. This is quite typical for ship global response. Some deck structures in the way of top superstructure showed equivalent stress values in the range from 10 to 80 MPa with no large deformations. This is because those deck structures are supported by bullheads and internal walls.

Results of the von-Mises stress highly depend on the mesh size. Thus, an area of interest, such as a large deck opening, may require mesh refinement. Class societies also give recommendations on the mesh sizing when the equivalent stress results are to be produced [51]. For the purposes of this thesis, the actual values considered to be of lower importance than the completed overall process of global strength analysis.

Since the mid ship section presents the highest stress concentration area, average equivalent stress results were produced amidships. Results for different frames are represented in Table 5.3 typical frame cross-sections are shown in Figure 5.12.

Table 5.3. Average equivalent stress

	Frame #124	**Frame #145**	**Frame #175**	**Whole model**
σ_e, [MPa]	72.4	88.4	65.8	58.8

As can be observed from the results, # 145experiences peak stresses. The last column of the table represents the overall average stress of the whole model, which tends to be lower than the values at the midship.

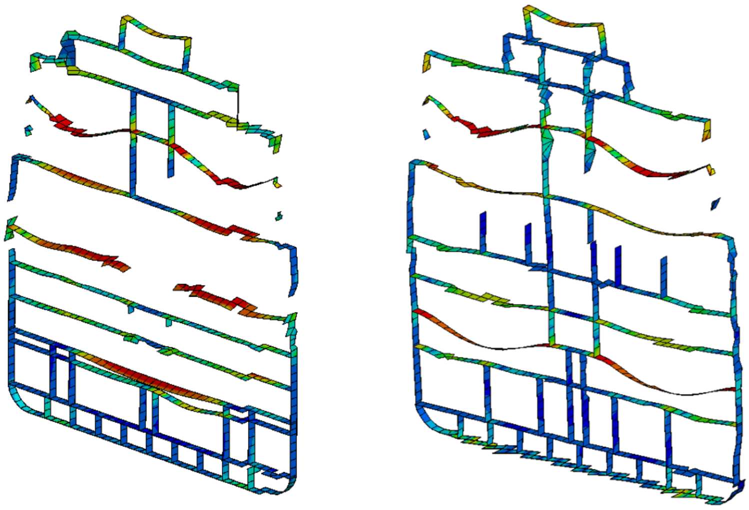


Figure 5.12. Equivalent stress σ_e distribution at frames 124 (left) and 145 (right)

5.4 Conclusions

The main aim of this thesis was to study the global strength analysis of a passenger ship in regular waves. The strength assessment procedure used state-of-the-art commercial software. However, because of practical technical problems associated with the CAD idealization software used to generate the FEA mesh some stiffening components of the model were ignored or over-simplified. Therefore, although the engineering principle followed are appropriate, some stress analysis results presented are not accurate.

A Quasi-dynamic approach was estimated to study the influence of fluid structure interaction on dynamic response. Accordingly, wave-induced hydrodynamic loading was mapped on the hull surface of the structural model.

The RAOs of ship motions,bending moments and shear forces were produced along with the added mass and radiation damping. Calculations were carried out for various headings, wave height and forward speed.

The vertical wave bending moment envelope obtained by the quasi-dynamic direct analysis procedure was compared against the Classification Society guidance on the maximum allowed values [51]. Comparison showed some differences that may be attributed to the inadequate strength stiffening of the global FEA model, mismatch of the longitudinal center of gravity and longitudinal center of flotation of the hydro-structural model, mass balancing and associated non-linear wave effects in hogging and sagging conditions. Thus, further validation of the results is left as future exercise.

To minimise uncertainties associated with this model in the future, the FEA model used should be reworked to adequately represent primary stiffeners in way of the decks. The model will also have to re-balanced.

A. RAO of ship motions

A.1 Surge, heave, pitch RAO at 0 forward speed

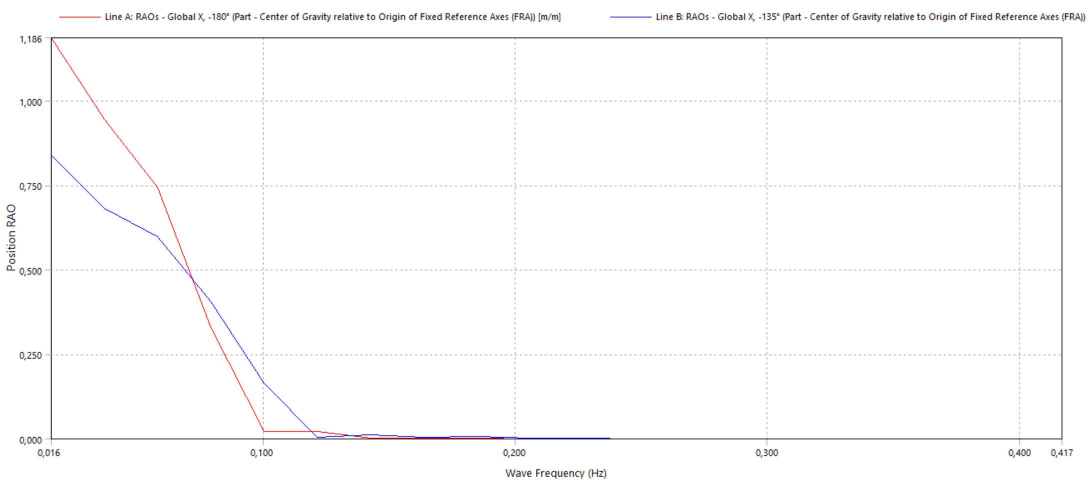


Figure 1.1. Surge at 180° and 135°

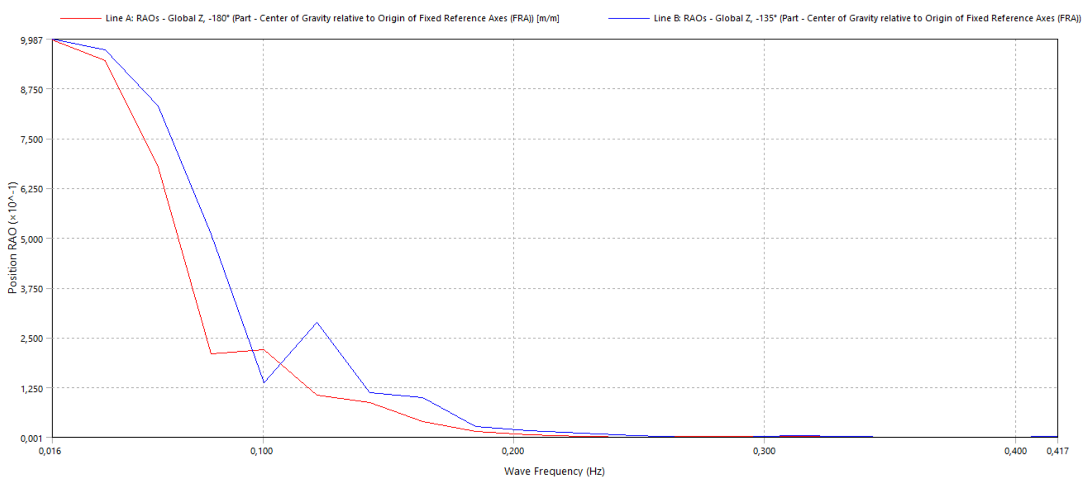


Figure 1.2. Heave at 180° and 135°

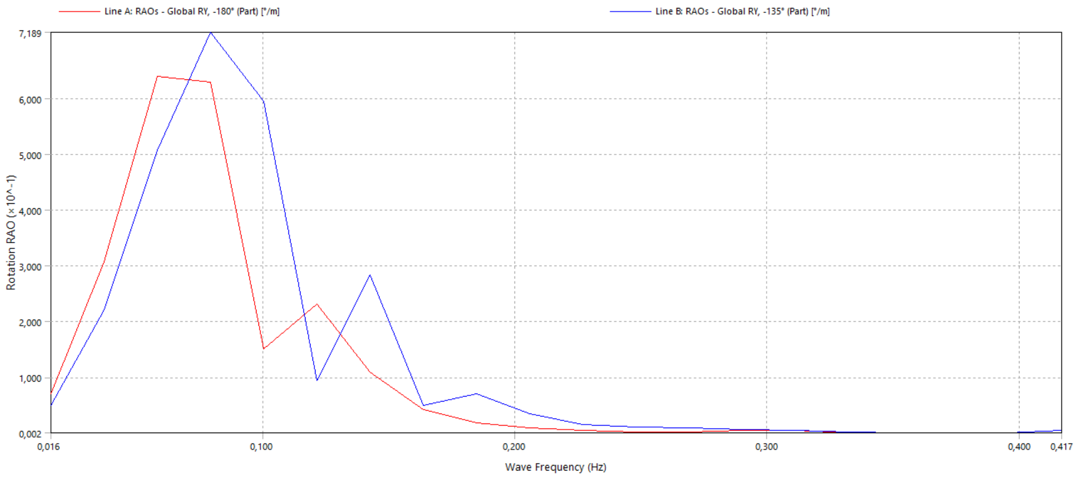


Figure 1.3. Pitch at 180° and 135°

A.2 Sway, roll, yaw RAO

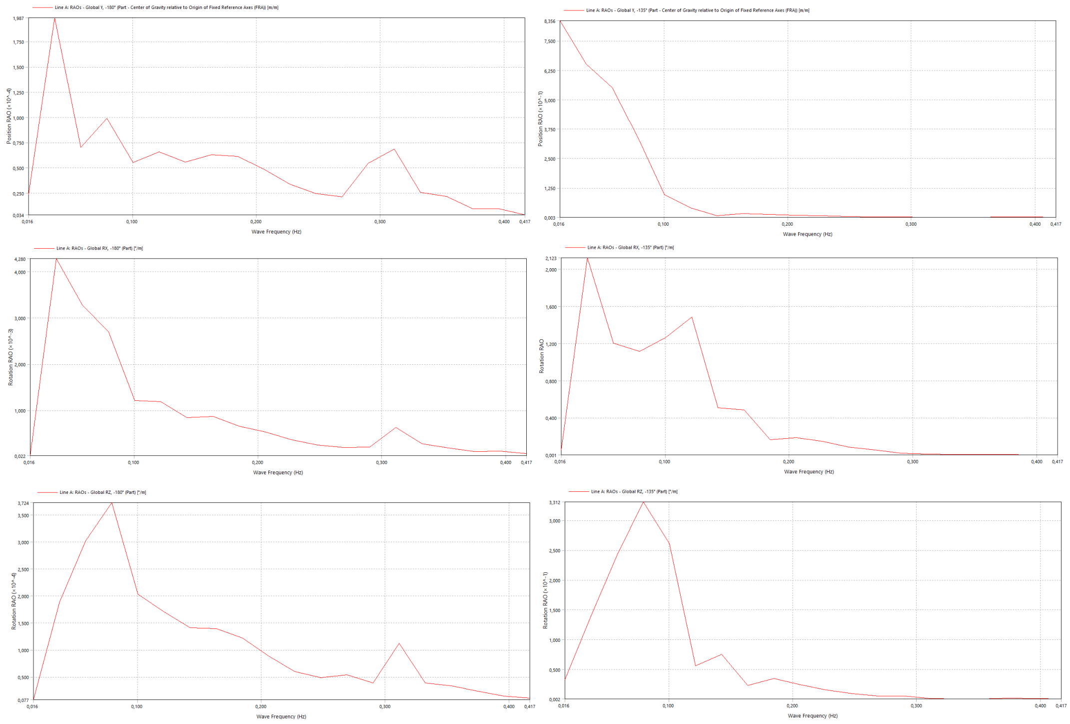


Figure 1.4. Sway, roll, yaw RAO at 180° (left) and 135° (right) at zero speed

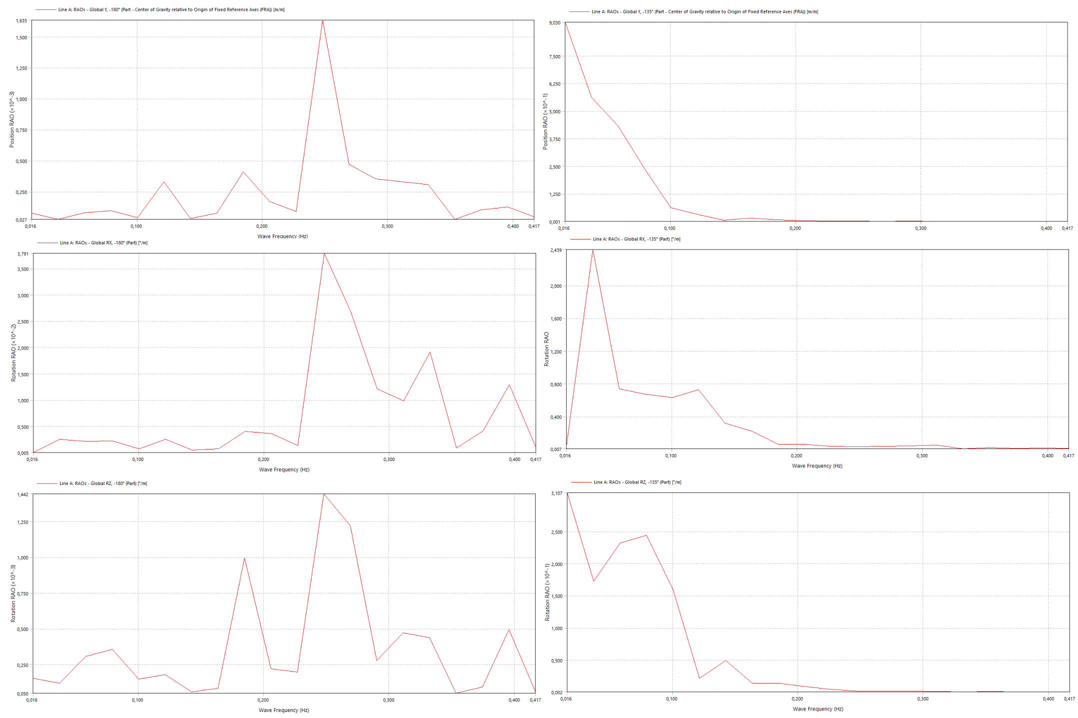


Figure 1.5. Sway, roll, yaw RAO at 180° (left) and 135° (right) at 15 kn speed

B. Added mass and radiation damping at 0 forward speed

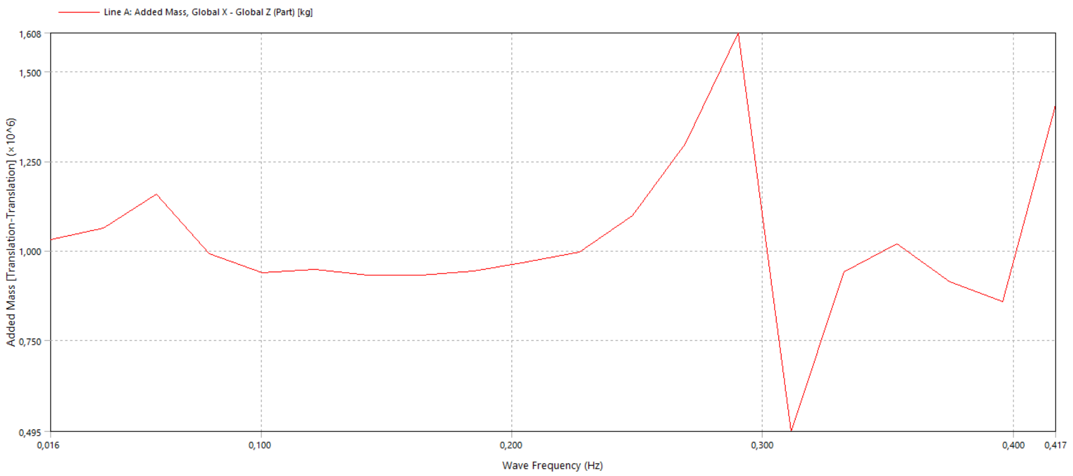


Figure 2.1. Added mass

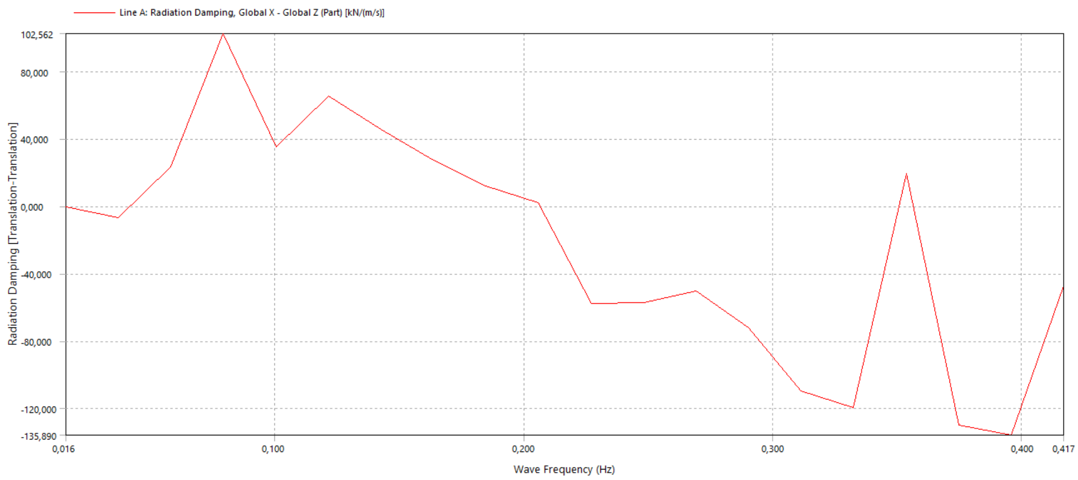


Figure 2.2. Radiation damping

C. Hydrodynamic pressures

C.1 Pressure at zero speed, 1 m wave, 180 and 135 degree headings

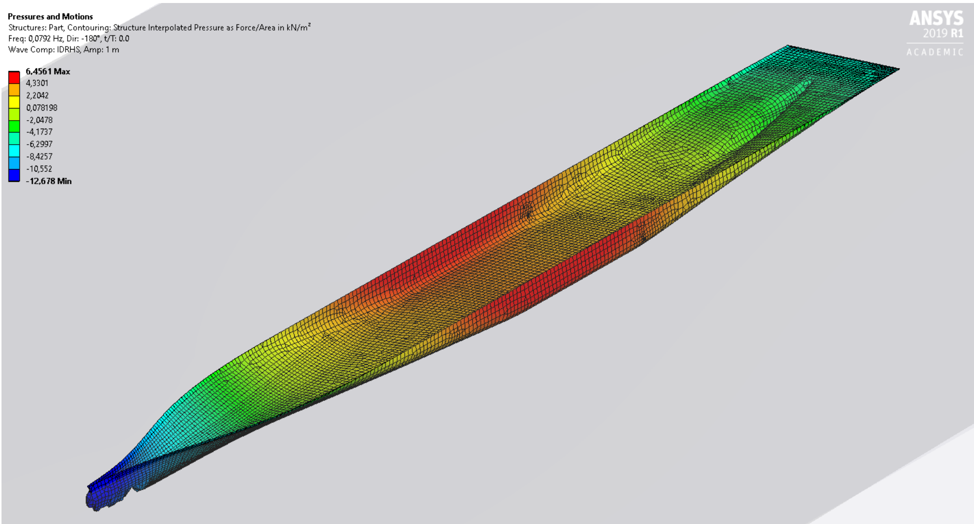


Figure 3.1. 180 degree heading

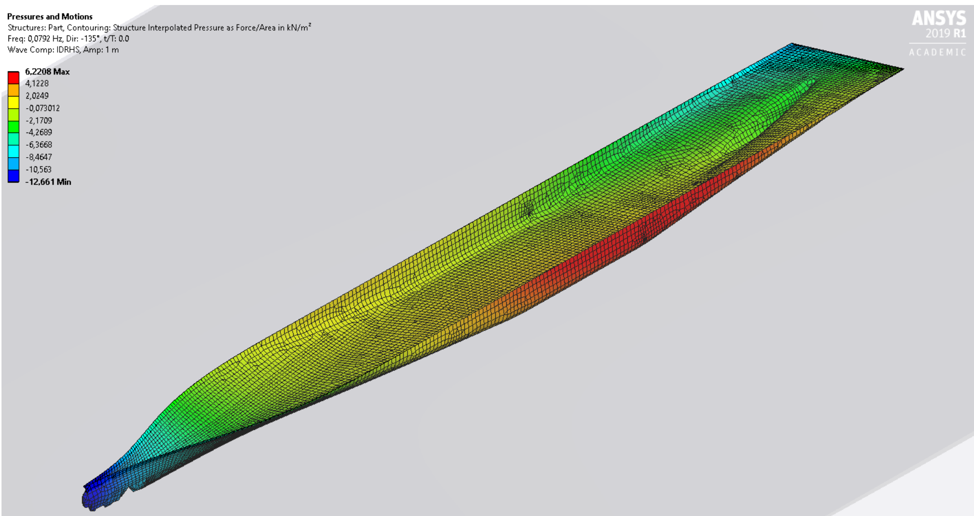


Figure 3.2. 135 degree heading

C.2 Pressure at zero speed 6 m wave, 180 and 135 degree headings

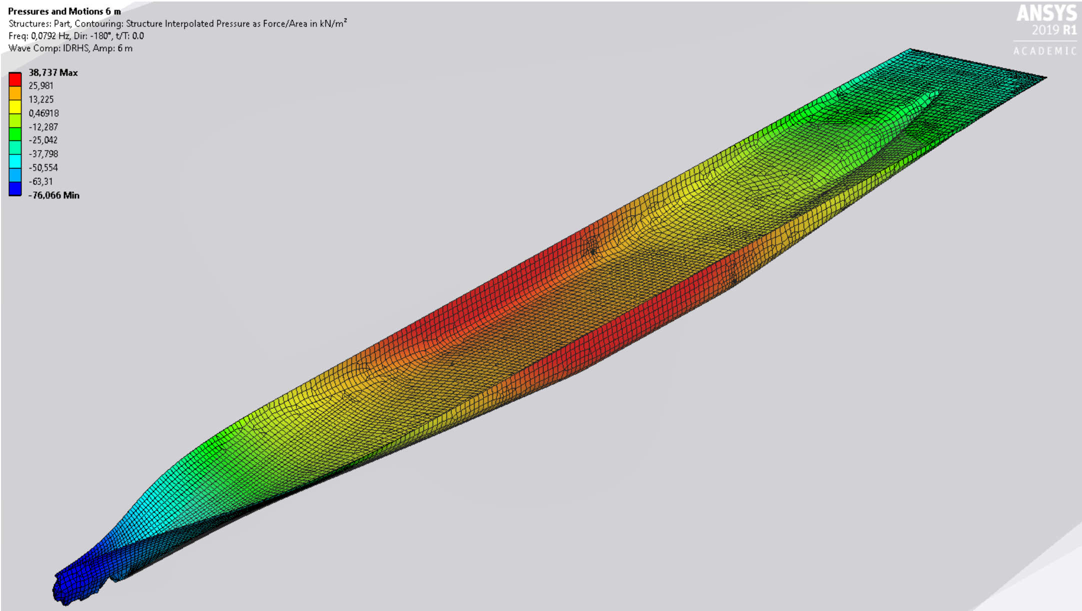


Figure 3.3. 180 degree heading

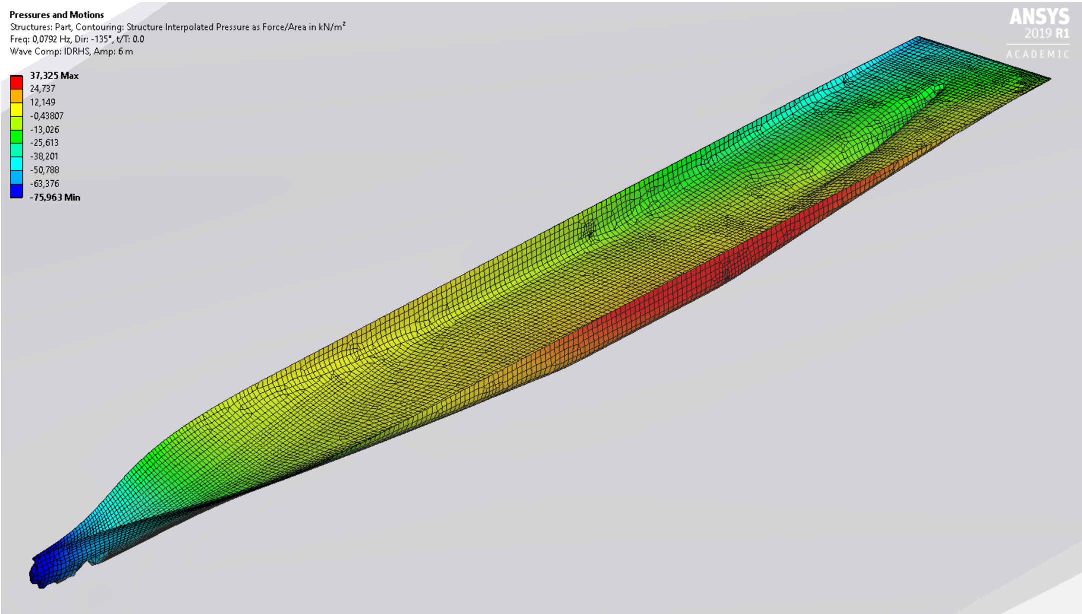


Figure 3.4. 135 degree heading

C.3 Pressure at 15 kn speed 1 m wave, 180 and 135 degree headings

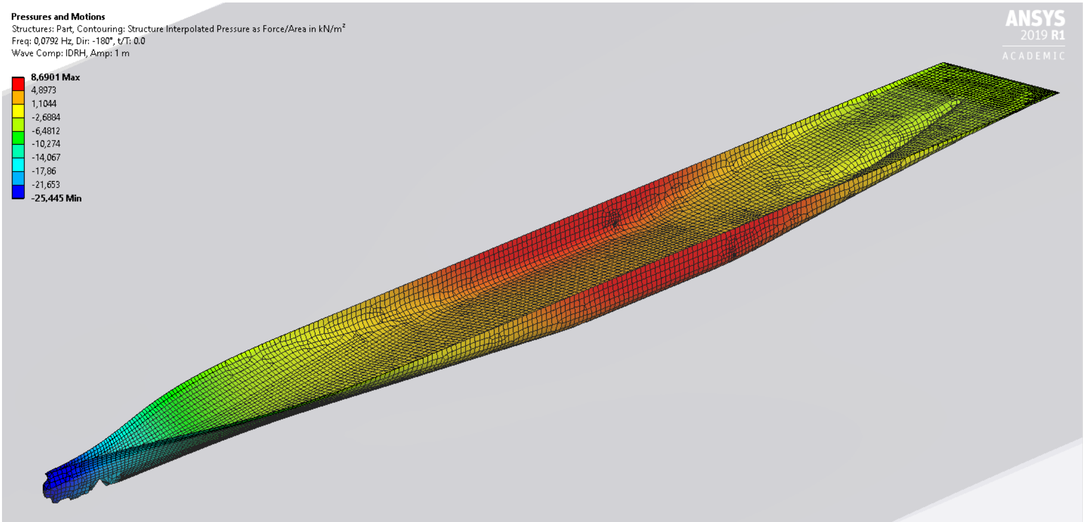


Figure 3.5. 180 degree heading

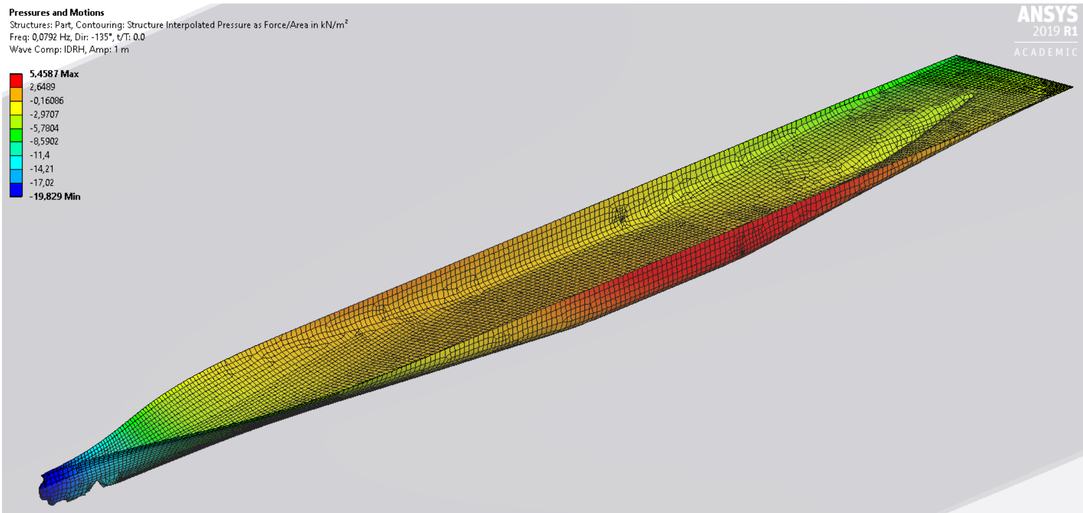


Figure 3.6. 135 degree heading

D. Conference material [1]

A rational quasi-dynamic FSI procedure for the evaluation of global loads used in the design of passenger ships

Nikita Dementyev[1,2,¥], Markus Jokinen[2], Antti Yrjänäinen[2], P.A.Lakshmarayanana[1] and Spyros Hirdaris[1]

[1]Aalto University, Maritime Technology Group, Espoo, Finland

[2]Elomatic Oy, Marine Consultancy and LCS, Espoo, Finland

[¥]Corresponding author (Email: nikita.dementyev@elomatic.com)

Abstract: The current boom in cruise and passenger ship markets has led to corresponding increase in the size of ships and their structural complexity. The optimization of capital expenditure costs remains a critical part in the design and construction of such ships. Additionally, to ensure sufficient functional safety margins the designers have at their disposal state-of-the-art tools and rational design methods for design and structural strength assessment especially for ships with general particulars and structural features that are not covered by the existing empirical Classification Rules.

This paper presents a rational quasi-dynamic response approach for the evaluation of global loads of passenger vessels. The method couples wave-induced hydrodynamic pressures with a rigid hull idealization performed with ANSYS AQWA and ANSYS SpaceClaim. The CAD structural model of a typical cruise ship was produced using CADMATIC Hull with basic design accuracy. Furthermore, model was transferred to ANSYS SpaceClaim to obtain an FEA model comprising of beam and shell elements representing the primary and secondary parts of the structure. NAPA software was used for evaluating the still water bending moment. Consequently, the 3D diffraction/radiation panel code ANSYS AQWA was used to define the wave pressures acting on the hull and loads were mapped on the hull surface and transferred to the ANSYS FEM solver for hydro-structure coupling. As a result, still water and wave bending moments are received as well as ship's response.

Comparisons against Class Society Rule wave bending moment and shear forces amidships demonstrates that the direct evaluation of the wave bending moment and shear force envelopes along the hull girder may be a preferred approach in terms of assuring global structural strength and optimizing total steel weight.

Key words: Passenger ships, global loads, quasi-dynamic response, FEA, ship hydrodynamics

Quasi-dynamic global strength analysis of a passenger ship in regular waves

Nikita Dementyev [1,2]
Markus Jokinen [1]
Antti Yrjänäinen [1]
Spyros Hirdaris [2]
P.A Lakshmarayanana [2]
[1] Elomatic Oy, Espoo
[2] Aalto University, Espoo
24.09.2019

Introduction

Aims

- Prepare the procedure and conduct global strength analysis for a typical cruise ship
- Implement quasi-dynamic analysis for global strength
- Compare class society's wave bending moment with the results

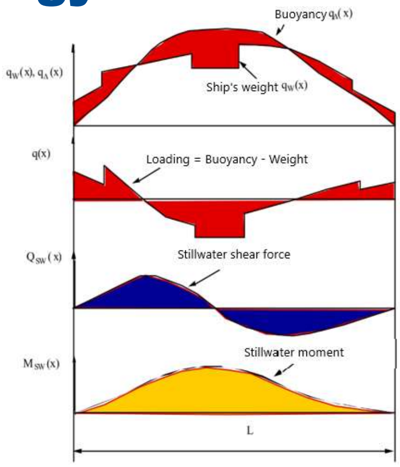
Limitations

- Conceptual early design level
- Attention to the hydrodynamic loading
- Simplified seakeeping analysis, for regular waves only

Methodology

1. **Still water condition**
   - Ship's weight
   - Buoyancy force = hydrostatic pressure

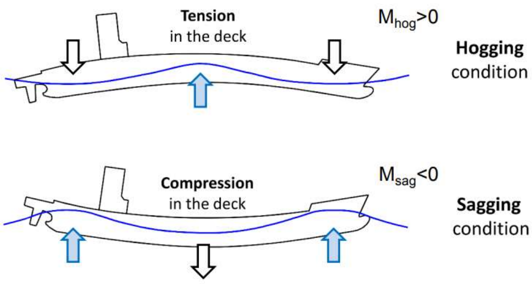
This results in:
   - Still water shear force
   - Still water bending moment



ELOMATIC

Methodology

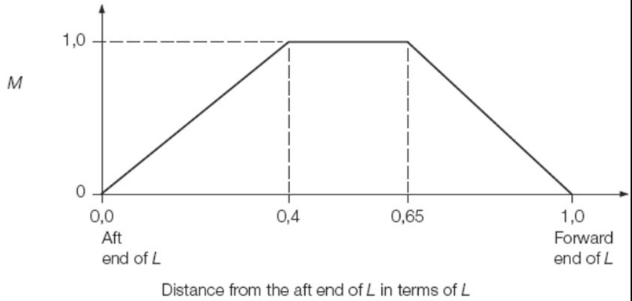
2. **Wave condition**
   - Vessel in waves + Still water loads
   - Wave-induced hydro-pressures
   - Higher hogging and sagging loads than in still water
   - Worst case scenario



ELOMATIC

Classification Society Approach

- Limitations for maximum BM, SF
- Values based on empirical statistics of vessel types
- Midship as the main interest area
- Direct analysis when cruise ship design limits are not covered by Class Rule BM,SF empirical formulae



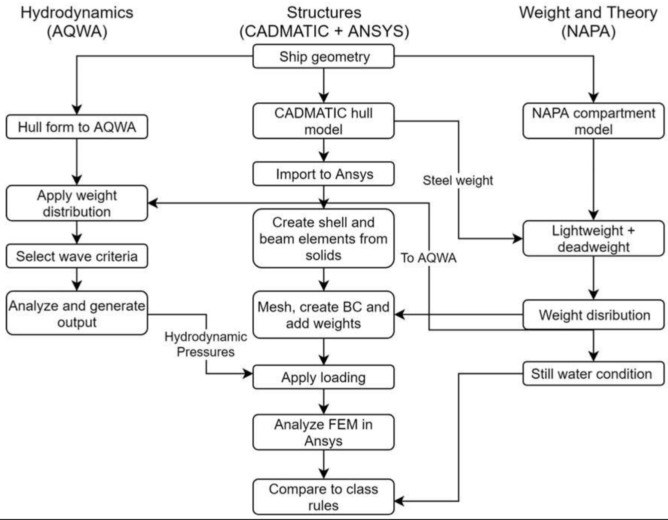
ELOMATIC

Quasi-static vs quasi-dynamic

	Quasi-static	Quasi-dynamic
Hydrodynamic loading	Static (Point load)	Dynamic (Hydrodynamic pressure)
Vessel in waves	Rigid	Rigid
Inertia effects	×	✓
Vibration effects	×	×
Structural analysis	Static	Static

ELOMATIC

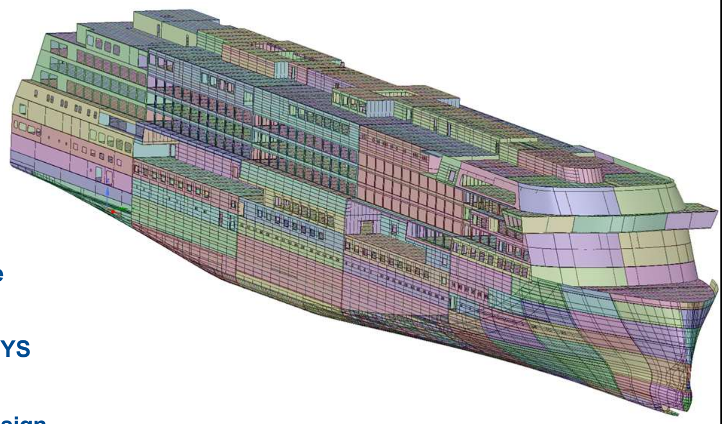
Modelling approach



ELOMATIC

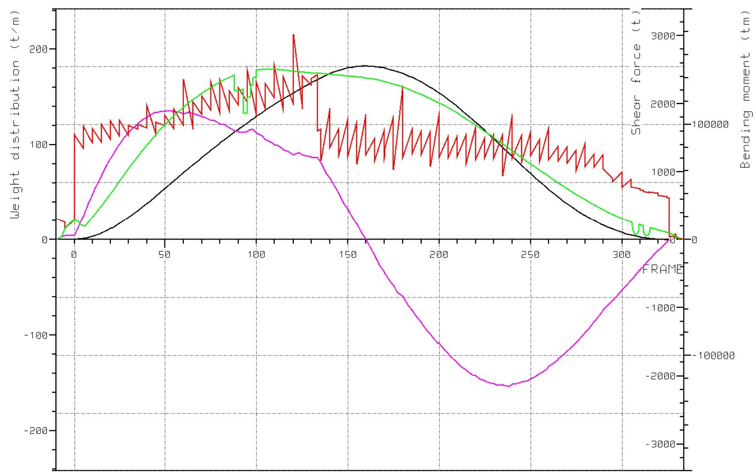
Case study vessel

- **A cruise ship**
- **Main Dimensions**
  - L = 240 m
  - B = 30 m
  - T_{Design} = 7 m
  - Δ = 32500 t
- **Model was initially made using CADMATIC hull**
- **Then transferred to ANSYS using .step**
- **Accuracy up to basic design**



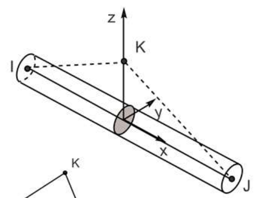
ELOMATIC

Still water condition



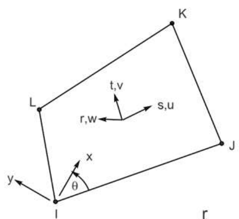
ELOMATIC

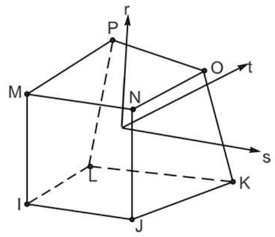
FE Modelling



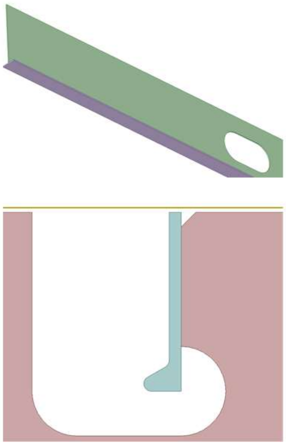
Three finite element idealizations were used :

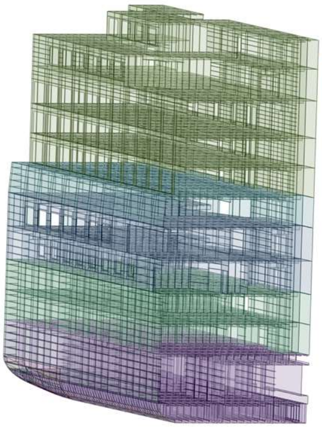
1. **Beam elements 2 or 3 nodes with 6 DOF**
   - Applicable for slender bodies
2. **Shell element (membrane effects)**
   - At least 3 nodes with 6 DOF
   - Used to model thin-walled bodies
3. **Solid element**
   - At least 4 nodes with 6 DOF
   - Used for complex geometry
   - When other element types are not suitable

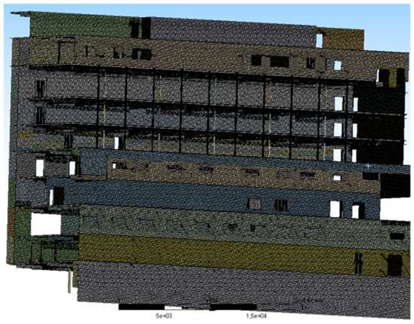




FEM model creation





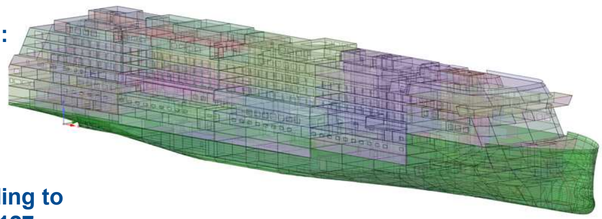


Modelling limitations & BC

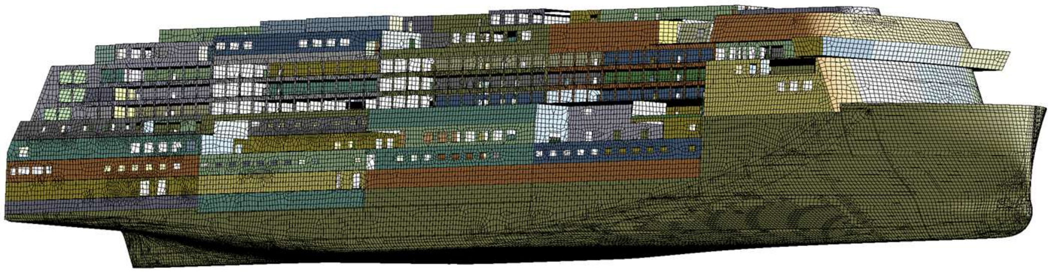
FEM model was heavily simplified:

- No stiffeners
- Only shell elements
- Equivalent plate method

Boundary Conditions (BC) according to DNVGL Classification Guideline 0127



FEM model



- Mesh size = 750 mm
- Nodes = 252 000
- Elements = 265 000

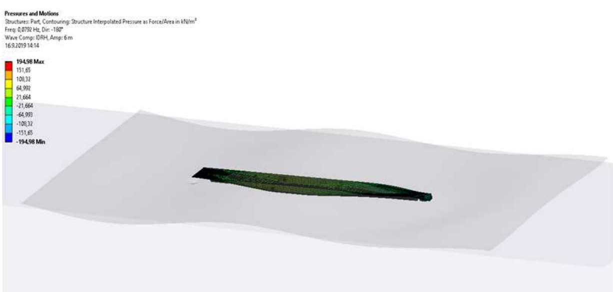
ELOMATIC

Hydrodynamic model

ANSYS AQWA:

- 3D panel method
- Simulation of wave diffraction and radiation forces
- Frequency domain with Green's function
- Regular wave analysis

Hydrodynamic pressures mapped to the structural model



ELOMATIC

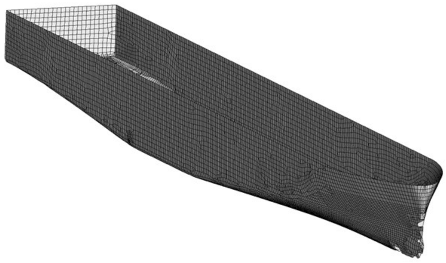
Ansys AQWA

Hydro model meshed with a coarse mesh

- Mesh size = 2 m
- COGz and moments of inertia as input
- Displacement and COG from hydrostatics

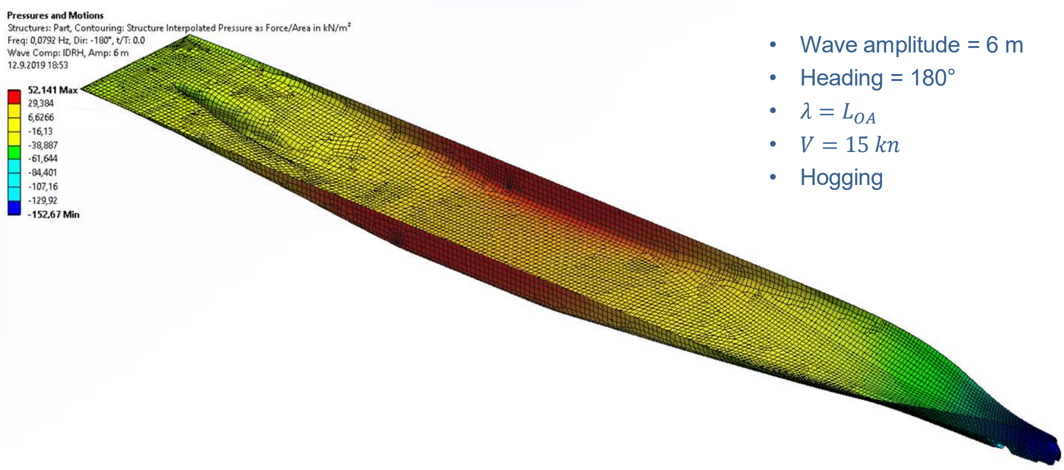
Hydrodynamic calculation carried out for:

- 0 and 15 kn forward speed
- 180° and 135 ° headings
- Wave frequencies from 0.015 Hz to 0.41 Hz
- Wave height 1 m and 6 m



ELOMATIC

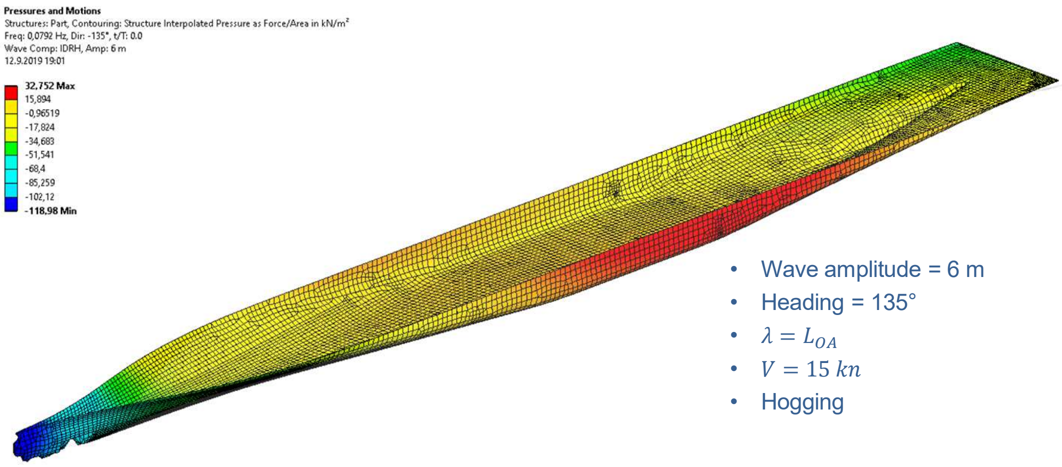
Hydrodynamic pressure



- Wave amplitude = 6 m
- Heading = 180°
- $\lambda = L_{OA}$
- $V = 15\ kn$
- Hogging

ELOMATIC

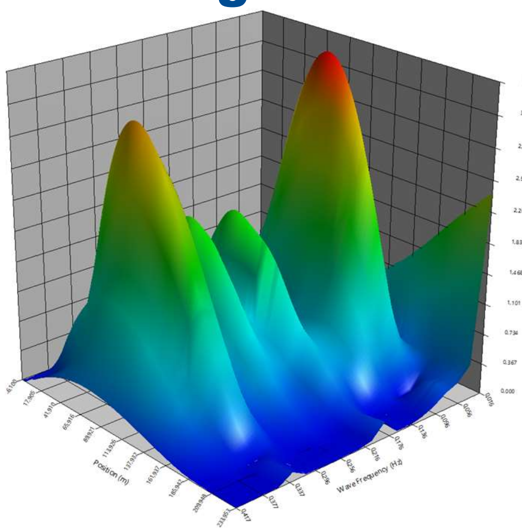
Hydrodynamic pressure

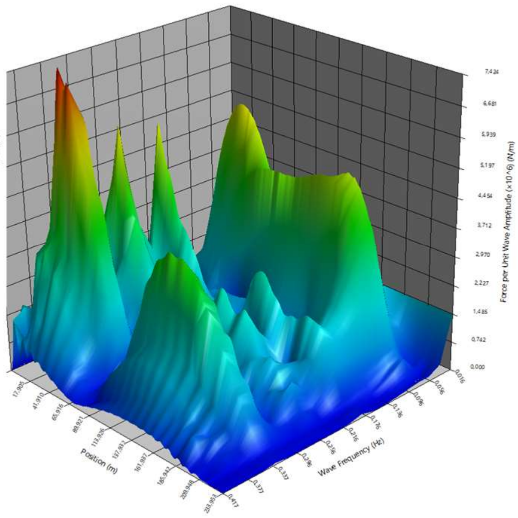


- Wave amplitude = 6 m
- Heading = 135°
- $\lambda = L_{OA}$
- $V = 15\ kn$
- Hogging

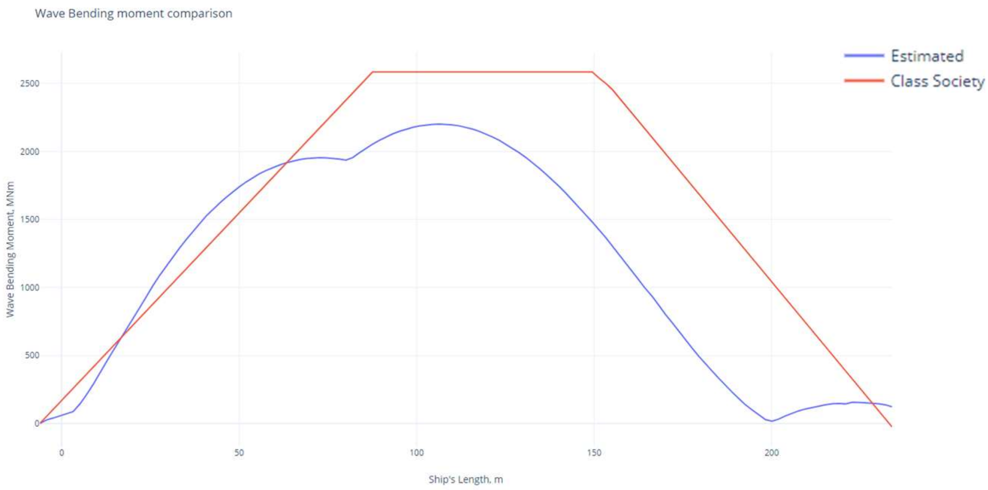
ELOMATIC

Bending moment and shear force RAO

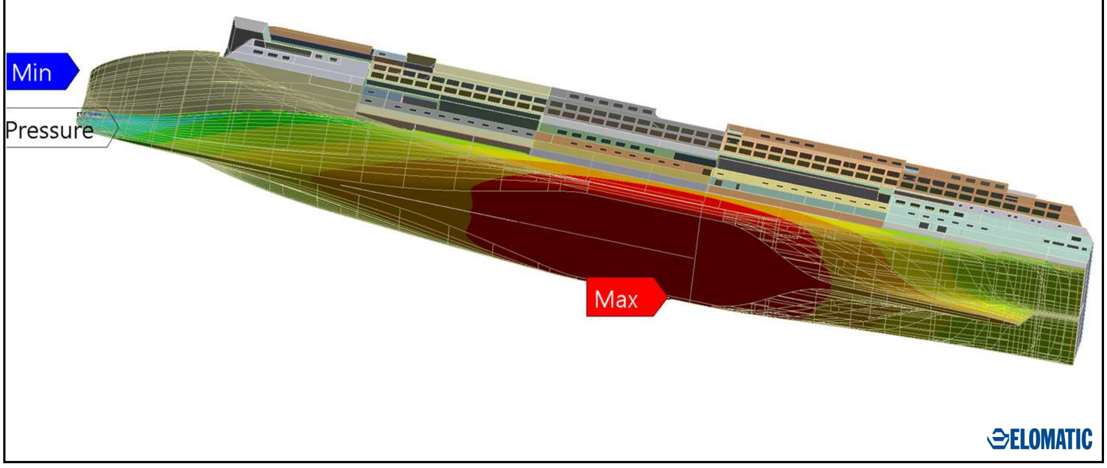




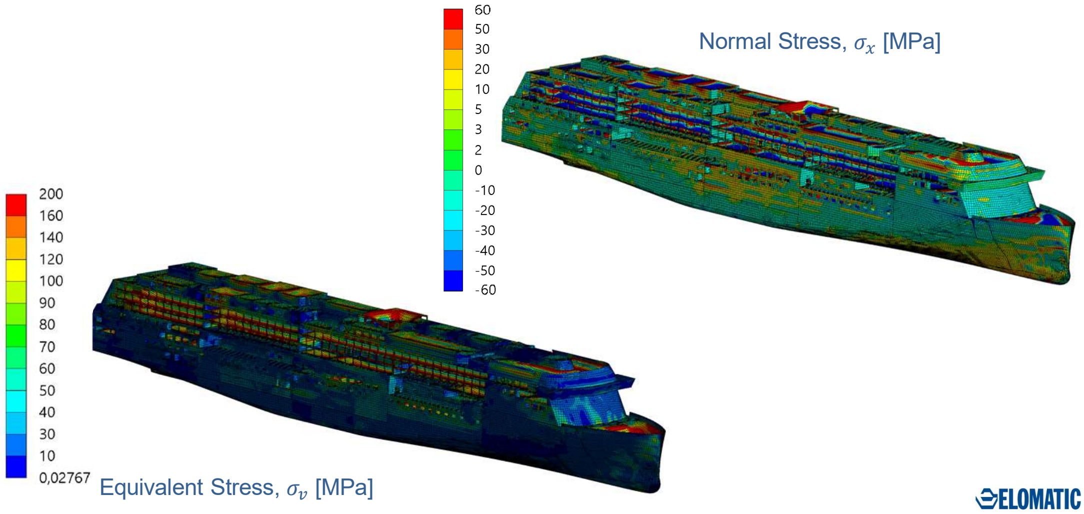
Wave bending moment



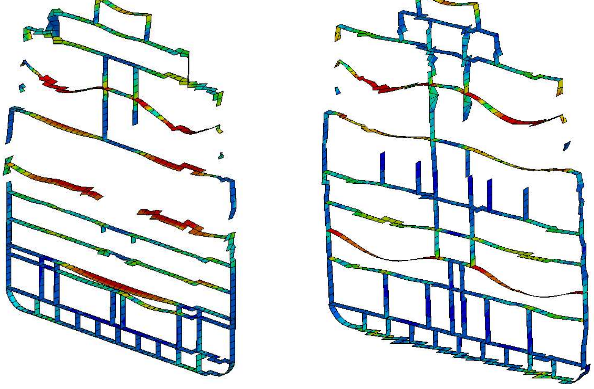
Mapped pressures



Stress results

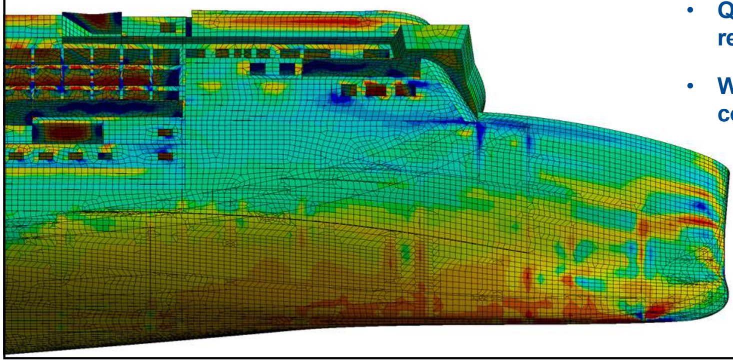


Average equivalent stress



	Frame #124	Frame #145	Frame #175	Whole model
σ_e, [MPa]	72.4	88.4	65.8	58.8

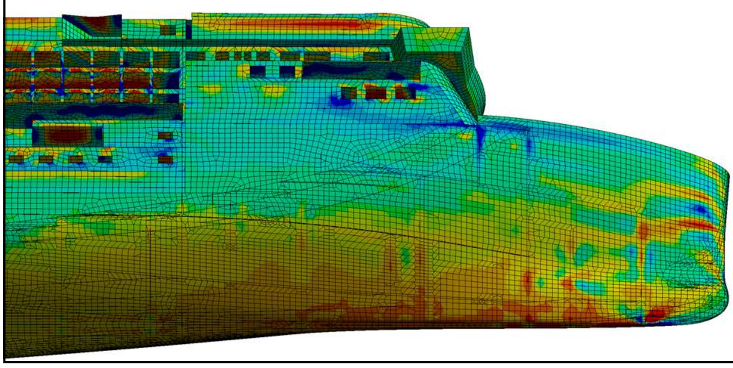
Conclusions



- **As a result global strength analysis procedure was carried out**
- **Quasi-dynamic approach used in regular waves**
- **Wave bending moment results compared to class rules**
  - Different outcomes using direct and class methods

ELOMATIC

Future development



- **Elaborate on the FEM model**
  - Fully involving equivalent plate method
  - Adding pillars
  - Adding weight elements and balancing models

Whole setup can later be used for fatigue or buckling estimation

ELOMATIC

Bibliography

[1] Turun Yliopisto, "The Baltic Seas International Maritime Conference – European Maritime Research from Adriatic to Baltic," 2019. [Online]. Available: https://meriteollisuus.teknologiateollisuus.fi/en/tapahtumat/baltic-seas-international-maritime-conference-european-maritime-research-adriatic-baltic

[2] Exploringmars.com, "Ship Broken In Half - Exploring Mars," 2017. [Online]. Available: http://www.exploringmars.org/ship-broken-in-half/

[3] O. F. Hughes and J. K. Paik, *Structural Analysis and Design*. Jersey City: The Society of Naval Architects and Marine Engineers, 2010, no. 1072.

[4] TheNavalArch.com, "Longitudinal Strength of Ships - an Introduction," 2015. [Online]. Available: https://thenavalarch.com/longitudinal-strength-ships-introduction/

[5] K. Rawson and E. Tupper, *Basic Ship Theory*, 5th ed. Oxford: Butterworth-Heinemann, 2001.

[6] DNV GL, "Class guidelines: Wave loads," Tech. Rep., 2018.

[7] MIT.EDU, "Marine Hydrodynamics," Tech. Rep., 2005. [Online]. Available: https://ocw.mit.edu/courses/mechanical-engineering/2-20-marine-hydrodynamics-13-021-spring-2005/lecture-notes/lecture22.pdf

[8] Altairuniversity.com, "Introduction to meshing," Tech. Rep., 2014. [Online]. Available: https://altairuniversity.com/wp-content/uploads/2014/02/meshing.pdf

[9] ANSYS Inc., "ANSYS Mechanical Help," 2019. [Online]. Available: https://ansyshelp.ansys.com

[10] J. Chaskalovic, "Strength of Materials," in *Mathematical Engineering*, 2014, no. 9783319035628, pp. 251–311.

[11] Lloyd's Register, "ShipRight. Design and Construction. Structural Design Assessment. Procedure for Primary Structure of Passenger Ships," Tech. Rep., 2017.

[12] ANSYS Inc., "AQWA User Manual," Tech. Rep., 2012.

[13] NZ Herald, "Coast under threat as more Rena containers overboard - NZ Herald," 2017. [Online]. Available: https://www.nzherald.co.nz/nz/news/article.cfm?c{_}id=1{&}objectid=10777406

[14] S. E. Hirdaris, W. Bai, D. Dessi, A. Ergin, X. Gu, O. A. Hermundstad, R. Huijsmans, K. Iijima, U. D. Nielsen, J. Parunov, N. Fonseca, A. Papanikolaou, K. Argyriadis, and A. Incecik, "Loads for use in the design of ships and offshore structures," *Ocean Engineering*, vol. 78, pp. 131–174, 2014.

[15] S. E. Hirdaris, N. J. White, N. Angoshtari, M. C. Johnson, Y. Lee, and N. Bakkers, "Wave loads and flexible fluid-structure interactions: Current developments and future directions," *Ships and Offshore Structures*, vol. 5, no. 4, pp. 307–325, 2010.

[16] C. G. Soares and Y. Garbatov, *Proceedings of the 19th International Ship and Offshore Structures Congress*, 2015.

[17] S. Hirdaris, Y. Lee, G. Mortola, A. Incecik, O. Turan, S. Hong, B. Kim, K. Kim, S. Bennett, S. Miao, and P. Temarel, "The influence of nonlinearities on the symmetric hydrodynamic response of a 10,000 TEU Container ship," *Ocean Engineering*, vol. 111, pp. 166–178, jan 2016. [Online]. Available: https://www.sciencedirect.com/science/article/pii/S002980181500596X?via{%}3Dihub

[18] L. Siipola, "Modern methods in ship structure analysis," 2018. [Online]. Available: https://blog.elomatic.com/en/modern-methods-in-ship-structure-analysis-

[19] S. Malenica and Q. Derbanne, "Hydro-structural issues in the design of ultra large container ships," *Int. J. Nav. Archit. Ocean Eng*, vol. 6, pp. 983–999, 2014.

[20] IACS, "Longitudinal Strength Standard," Tech. Rep. July, 2016.

[21] DNV GL, "Class guideline: Direct strength analysis of hull structures in passenger ships," Tech. Rep., 2016.

[22] Lloyd's Register, "ShipRight Design and construction - Additional Design Procedures," no. July, 2014.

[23] J. Rorup, T. E. Schellin, and H. Rathje, "Load Generation for Structural Strength Analysis of Large Containerships," *Volume 2: Structures, Safety and Reliability*, no. September 2016, pp. 89–98, 2008.

[24] H. G. Payer and T. E. Schellin, "A class society's view on rationally based ship structural design," *Ships and Offshore Structures*, vol. 8, no. 3-4, pp. 319–336, 2013.

[25] T. E. Schellin and O. el Moctar, "Numerical Prediction of Impact-Related Wave Loads on Ships," *Journal of Offshore Mechanics and Arctic Engineering*, vol. 129, no. 1, pp. 39–47, feb 2007.

[26] H. Naar, "Ultimate strengh of hull girder for passenger ships," Ph.D. dissertation, Helsinki University of Technology, 2006. [Online]. Available: https://aaltodoc.aalto.fi/handle/123456789/2669

[27] Sener, "FORAN Marine," 2019. [Online]. Available: http://www.marine.sener/foran

[28] AVEVA, "AVEVA Marine," 2019. [Online]. Available: https://sw.aveva.com/marine

[29] NAPA, "Solutions for Ship Design," 2019. [Online]. Available: https://www.napa.fi/software-and-services/ship-design/

[30] CADMATIC, "Marine - CADMATIC," 2019. [Online]. Available: https://www.cadmatic.com/marine

[31] Bureau Veritas, "MARS 2000 2D ship structural assessment software," 2019. [Online]. Available: https://marine-offshore.bureauveritas.com/mars-2000-2d-ship-structural-assessment-software

[32] DNV GL, "Nauticus Hull - DNV GL," 2019. [Online]. Available: https://www.dnvgl.com/services/ship-structural-analysis-and-design-nauticus-hull-1061

[33] Lloyd's Register, "RulesCalc from Lloyd's Register," 2019. [Online]. Available: https://www.lr.org/en/rulescalc/

[34] Bureau Veritas, "Hydrostar software," 2019. [Online]. Available: https://marine-offshore.bureauveritas.com/hydrostar-software-powerful-hydrodynamic

[35] Lloyd's Register, "Waveload-FD," 2019. [Online]. Available: http://www.webstore.lr.org/products/2858-waveload-fd-12-months-subscription-licence.aspx

[36] 3ds Simulia, "Abaqus Unified FEA," 2019. [Online]. Available: https://www.3ds.com/products-services/simulia/products/abaqus/

[37] Siemens PLM, "Femap," 2019. [Online]. Available: https://www.plm.automation.siemens.com/global/en/products/simcenter/femap.html

[38] DNV GL, "Strength assessment of hull structures – POSEIDON - DNV GL," 2019. [Online]. Available: https://www.dnvgl.com/services/strength-assessment-of-hull-structures-poseidon-18518

[39] Lloyd's Register, "LR ShipRight procedures," 2019. [Online]. Available: https://www.lr.org/en/shipright-procedures/{#}accordion-shiprightoverview

[40] Wärtsilä, "Deadweight," 2017. [Online]. Available: https://www.wartsila.com/encyclopedia/term/deadweight-(dwt)

[41] T. Kukkanen, "Wave load predictions for marine structures," *Journal of Structural Mechanics)*, vol. 43, no. 3, pp. 150–166, 2010.

[42] S. E. Hirdaris, "Wave loads - developments and directions," *The Naval Architect*, 2010.

[43] N. Kornev, "Ship dynamics in waves," Faculty of Mechanical Engineering and Marine Technology Chair of Modelling and Simulation, Rostock, Tech. Rep., 2012.

[44] ITTC, "ITTC-Recommended Procedures and Guidelines Numerical Estimation of Roll Damping," Tech. Rep., 2017. [Online]. Available: https://www.ittc.info/media/8151/75-02-07-045.pdf

[45] F. cae engineering, "Element Types," 2014. [Online]. Available: http://fea-cae-engineering.com/fea-cae-engineering/element{_}types.htm

[46] SpaceClaim Help, "Shared topology in ANSYS," 2019. [Online]. Available: http://help.spaceclaim.com/2015.0.0/en/Content/ANSYS{_}SharedTopology.htm

[47] DNV GL, "Class guideline: Finite element analysis," Tech. Rep. October, 2015.

[48] ——, "Rules for Classification: Hull. Finite element analysis," Tech. Rep., 2018.

[49] J. Matusiak, *Laivan Kelluvuus ja Vakavuus*, Espoo, 1995.

[50] ANSYS Inc., "AQWA Theory Manual," 2019. [Online]. Available: https://ansyshelp.ansys.com/

[51] DNV GL, "Rules for classification. Ships: Structural strength and integrity," Tech. Rep., 2018.